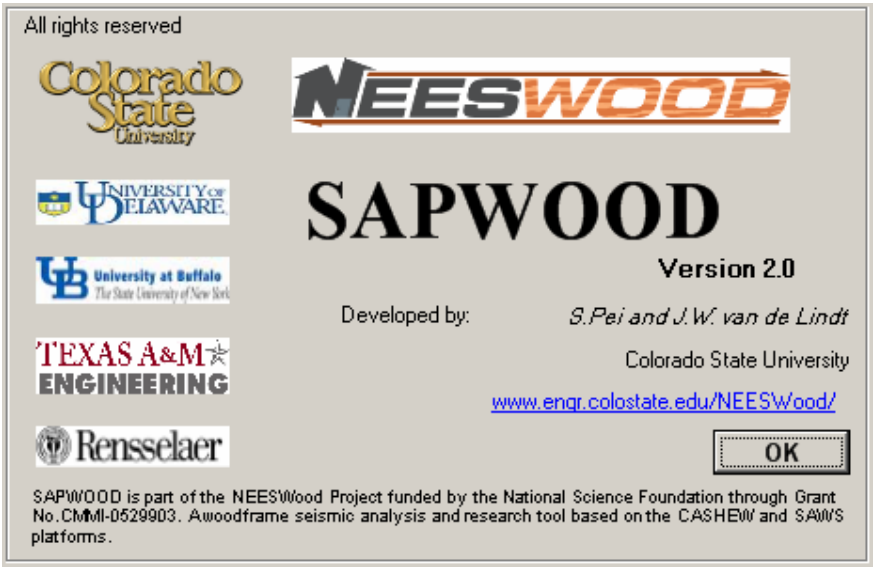
User's Manual For

SAPWood for Windows

Seismic Analysis Package for Woodframe Structures

Version 2.0


All rights reserved
Colorado State University
NEESWOOD
SAPWOOD
Version 2.0
Developed by:
S.Pei and J.W. van de Lindt
Colorado State University
www.engr.colostate.edu/NEESWood/
OK
SAPWOOD is part of the NEESWood Project funded by the National Science Foundation through Grant No.CMMI-0529903. A woodframe seismic analysis and research tool based on the CASHEW and SAWS platforms.

By

Shiling Pei & John W. van de Lindt

Colorado State University

2010

Free download available at:
http://www.engr.colostate.edu/NEESWood/SAPWood.htm

ACKNOWLEDGEMENT

Funding for this study was provided through the National Science Foundation's George E. Brown, Jr. Network for Earthquake Engineering Research (NEES-R) program via NSF grant CMMI-0529903. That funding is gratefully acknowledged.

DISCLAIMER

Opinions and results incorporated in this software package are those of the authors. Although the software is believed to be accurate, any computational results from use of this software must be verified by the use prior to use in design. The authors hold no liability for any consequences resulting from application of this software to the design and/or construction of structural systems or components. No liability for the information included in this software package is assumed by National Science Foundation or NEESInc.

NEESWOOD PROJECT OVERVIEW

The NEESWood Project is a four-year five-university project headquartered at Colorado State University. The project director is John W. van de Lindt at Colorado State University (now at University of Alabama), and Co-Investigators are Rachel Davidson at the University of Delaware, Andre Filiatrault at the University at Buffalo, David V. Rosowsky at Texas A&M University, and Michael Symans at Rensselaer Polytechnic Institute.

The objective of the NEESWood Project is to develop a logical and economical performance based seismic design philosophy to safely increase the height of woodframe construction in regions of moderate to high seismicity. For further details please see the project website at http://www.engr.colostate.edu/NEESWood/.

Table of Contents

Section	Page
1 Overview	1
1.1 What is SAPWood	1
1.2 SAPWood Components	1
2 Tools and usage	3
2.1 Main Frame	3
2.2 Model Builder	5
2.3 Single Earthquake Excitation	9
2.4 Incremental Dynamic Analysis (IDA)	14
2.5 SDOF System Identification	16
2.6 Multi-Record IDA	18
2.7 Manual Fitting tool	19
2.8 SAPWood-Nail Pattern	20
2.8.1 NP Model Builder	21
2.8.2 NP Analysis	23
2.9 Wall parameter database	25
2.10 Loss analysis	26
2.10.1 Vulnerability based loss simulation	27
2.10.2 Virtual Hazard simulation	30
3 Trouble shooting	31
3.1 Can't launch the program	31
3.2 Fail to load model file	32
3.3 Unexpected termination during analysis	32
3.4 Unexpected termination during plotting	32
3.5 Unexpected error during file saving	32
4 FAQs	33
5 Examples	37
5.1 Benchmark structure analysis	37
5.1.1 Building the model	37
5.1.2 Single earthquake excitation analysis	40
5.1.3 IDA analysis	45
5.1.4 SDOF Identification analysis	47
5.2 SAPWood-NP wall model analysis	49
5.2.1 NP model building	49
5.2.2 NP model analysis	51
5.3 Woodframe structure loss analysis	53
References	56
Appendix	58
A. Input file formats	58
B. Output file formats	64
C. Spring models	71

D. Example structure configuration 75

1 Overview

1.1 What is SAPWood

The program discussed in this manual is Version 2.0 of the Seismic Analysis Package for Woodframe structures (SAPWood). It is now available for free download from the NEESWood SAPWood webpage at:

http://www.engr.colostate.edu/NEESWood/SAPWood.htm

This analysis program, developed based on the Seismic Analysis of Woodframe Structures (SAWS) and Computer Program for the Cyclic Analysis of SHEar Walls (CASHEW) (see Folz and Filiatrault, 2002) concepts, is aimed at providing both researchers and practitioners with a user-friendly software package which is capable of performing nonlinear seismic structural analysis and loss analysis for woodframe structures. In addition to time domain analysis, several modules that support the NEESWood PBSD efforts are included and described in Section 1.2. This software package is used extensively in the NEESWood project and may have upgrade versions depending on the advancement in analysis models for light frame wood structure. If you wish to be informed about the future upgrades, please send an email to the authors (John W. van de Lindt at jwvandelindt@engr.ua.edu and Shiling Pei at shiling.pei@sdstate.edu).

1.2 SAPWood Components

SAPWood is essentially a toolbox which allows the user to build, load, modify, and save a light frame wood structural model for various seismic-related analyses. There are three types of models currently available in SAPWood: the first one is a bi-axial structural model which was introduced by Folz and Filiatrault (2002) in the SAWS program (3DOF in each story with a rigid diaphragm assumption); the second one is a simplified lumped-mass shear building model (with only 1DOF at each story level), which can be useful for preliminary uni-directional analysis and simplified design approaches; and a fully coupled bi-axial shear and bending model (triaxial model, Pei and van de Lindt, 2009, van de Lindt et al. 2010). The triaxial model also has the option for users to add friction pendulum base isolators to the base level. SAPWood also provides the user the ability to build and analyze light frame wood shearwalls using nonlinear connectors (nails, hold-down devices, screws, etc.) elements. This enables the analysis of woodframe structures beginning at the fastener level when assembly (shearwall) test data is not available. The module termed SAPWood-Nail Pattern (NP) analysis is designed to perform this task (the concept is similar to the CASHEW program developed in the CUREE-Caltech project with some modifications). SAPWood also provides the option of obtaining the shearwall and drywall component parameters directly from a database that is provided with the program. However, the parameters from the database should only be used to conduct preliminary trial analysis since the variation of real shearwall behavior cannot be reflected by the fixed database. Also included in Version 2.0 of SAPWood is a preliminary loss estimation module, in which vulnerability analysis and long term loss simulation can be performed to evaluate the economic loss of a structure during earthquakes. The loss analysis is limited to biaxial models only.

The analysis options currently in SAPWood include: Traditional nonlinear time domain earthquake

excitation, Incremental Dynamic Analysis (IDA), System (hysteresis) Identification, Multi-case IDA, assembly level analysis (SAPWood-NP), a shear wall parameter database, vulnerability-based loss simulation, and Virtual Hazard loss simulation. Multi-case IDA can be used as a batch analysis tool to perform analyses with a combination of multiple structural systems, earthquake records, and intensity levels. This is useful for the comparison when competing designs are being considered. There is also a fitting tool within SAPWood that provides the user with semi-automatic fitting of shearwall test hysteresis data for any one of the four possible spring models that can be used in a SAPWood structural model.

Currently, there are four spring models included in the SAPWood package: linear spring model, bilinear spring model, SAWS-type ten-parameter hysteretic model ("CUREE model" from CUREE-Caltech project), and a 16-parameter evolutionary parameter hysteretic model ("EPHM model"). The detailed information of these models can be found in the appendix of this document.

2 Tools and usage

This section explains the purpose and provides instruction for each analysis tool in SAPWood.

2.1 Main Frame

Purpose: This window allows one to load a structure and/or different tools into the SAPWood workspace (memory).

The Main Frame has three parts, the Main Menu, Workspace, and Environment. The Main Menu is where the model and all the tools in SAPWood can be loaded for use. SAPWood currently supports House models (*.sap or *.dat) and Shear Building models (*.sbd). SAWS input files (*.dat) can be directly uploaded into SAPWood and the model is auto generated (however the wall length and location will not be uploaded since this information is not in a SAWS input file). The Workspace provides a place to view and modify the structural models just loaded into memory (note that a user may load as many models as he/she wishes). The Environment allows users to modify the Unit system (i.e. Metric or U.S. Customary) used in the calculation based on the unit system in the input model file. It also allows the user to change the path for all file outputs from SAPWood. However, note that the default output is set to the root directory of the C: drive (C:\). The PGA/Sa option in the environment allows the user to use spectral acceleration (Sa) or peak ground acceleration (PGA) in order to specify the intensity/magnitude of the earthquake for all the SAPWood tool inputs (except loss estimation, where the choice of the intensity indicator is specified in the earthquake environment information file). If the Sa option is used, the user must also specify the target period (Tn) and damping ratio at which the Sa is to be evaluated.

Note: Some of the modules have not changed from V1.0 to V2.0. Thus part of the figures uses the V1.0 display.

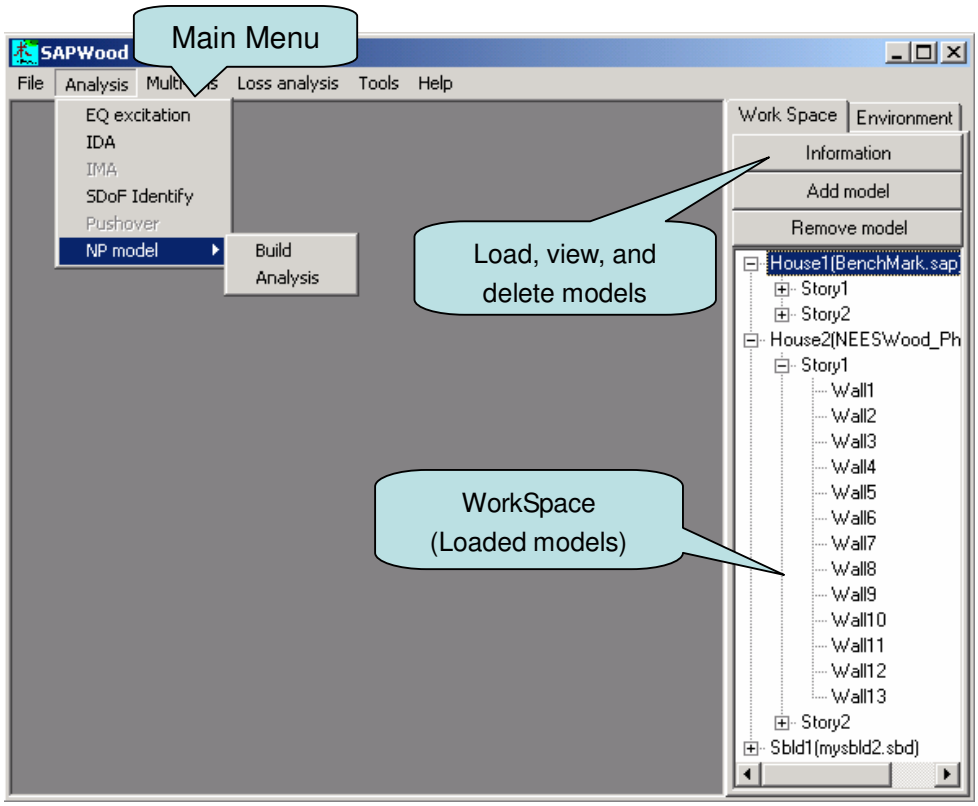


Figure 1. Main frame and work space

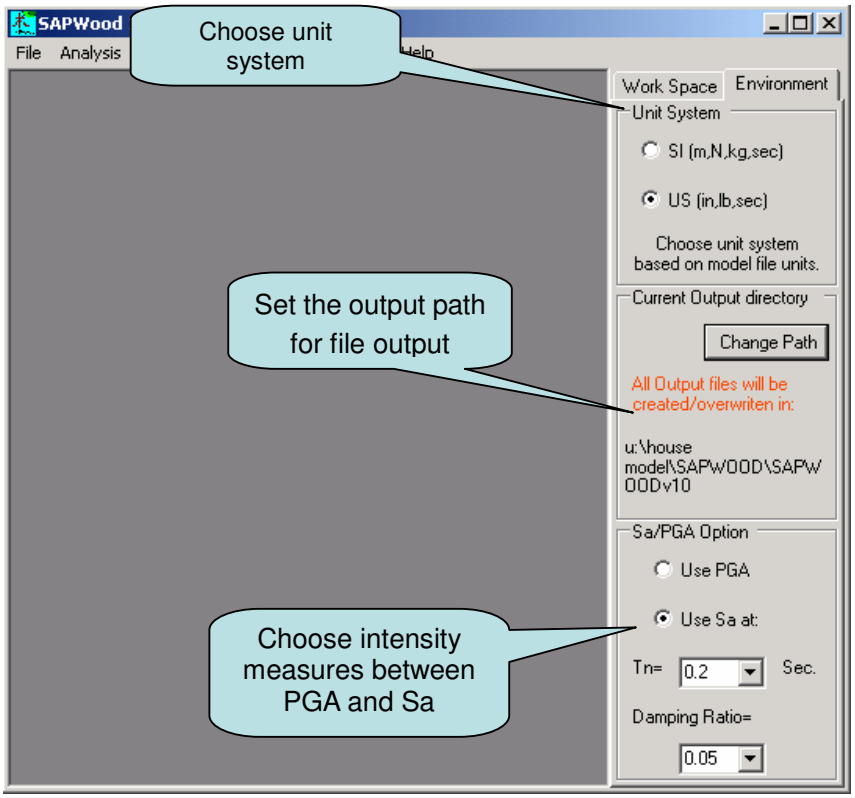


Figure 2. Main frame and environment

Tips:

- It is suggested that upon loading SAPWood you change the output directory setting to the place you would like your files saved. Otherwise, all the output files (including the model file constructed by/within the model builder) will be output to C:\ by default.
- All files will be OVERWRITTEN without warning if you use the same file name and directory.
- There are two ways to upload a model; one is from File-> Load model, the other is from Workspace-> Add model. There is no difference between them.
- You can load multiple models into the workspace, they will be automatically named as HouseX(filename) or SbldX(filename) depending on whether it is a biaxial house model or a shear building model. X denotes the sequence of loading. The target model file can be viewed by selecting the model and clicking "Information".
- You can bring up the property window (which will be introduced later) and view the information for a particular model simply by selecting the (story, wall) and clicking on "Information". You can also alter the values by clicking "EDIT mode" on the property windows (for story and wall), changing the value, and clicking "Update" on the property windows. The model in memory will be changed, but the model file will NOT change.
- SAPWood will NOT automatically convert units. The Units option in the *Environment* simply tells the program what the units are for the model input file. Then the program uses the proper value for gravity in the calculation.
- If the Sa option in the environment tab is selected, all earthquake scaling within most of the analysis tools in SAPWood will be performed based on Sa. The program will automatically call a subroutine to scale the earthquake to the desired Sa based on the input values for both Tn and the associated damping ratio.

2.2 Model Builder

Purpose: To provide a tool to build, modify, and save the structural models (biaxial model and shear building model only. Triaxial model needed to be build in a text editor) used in SAPWood.

It is highly recommended that the use construct the model input file using text editor following the format explained in the "Model file format instruction.xls" file distributed with this package. For a complicated model, using the interactive model builder may be less efficient than directly create model files.

There are two tabs in the model builder: one for biaxial house model and one for shear building model. The user starts out by assigning the "Global" information (total number of stories and number of structural walls in each story for the model; or total number of stories for a shear building) and sets up an empty frame. Then the user can double-click on the component she/he wants to define and bring up the property window. In the property window, the properties (spring type, parameters, mass, shape… etc.) can be set and updated. After all component properties have been assigned, the model can be saved to the output folder with the desired name. This procedure is illustrated in Figure 3 with 5 steps:

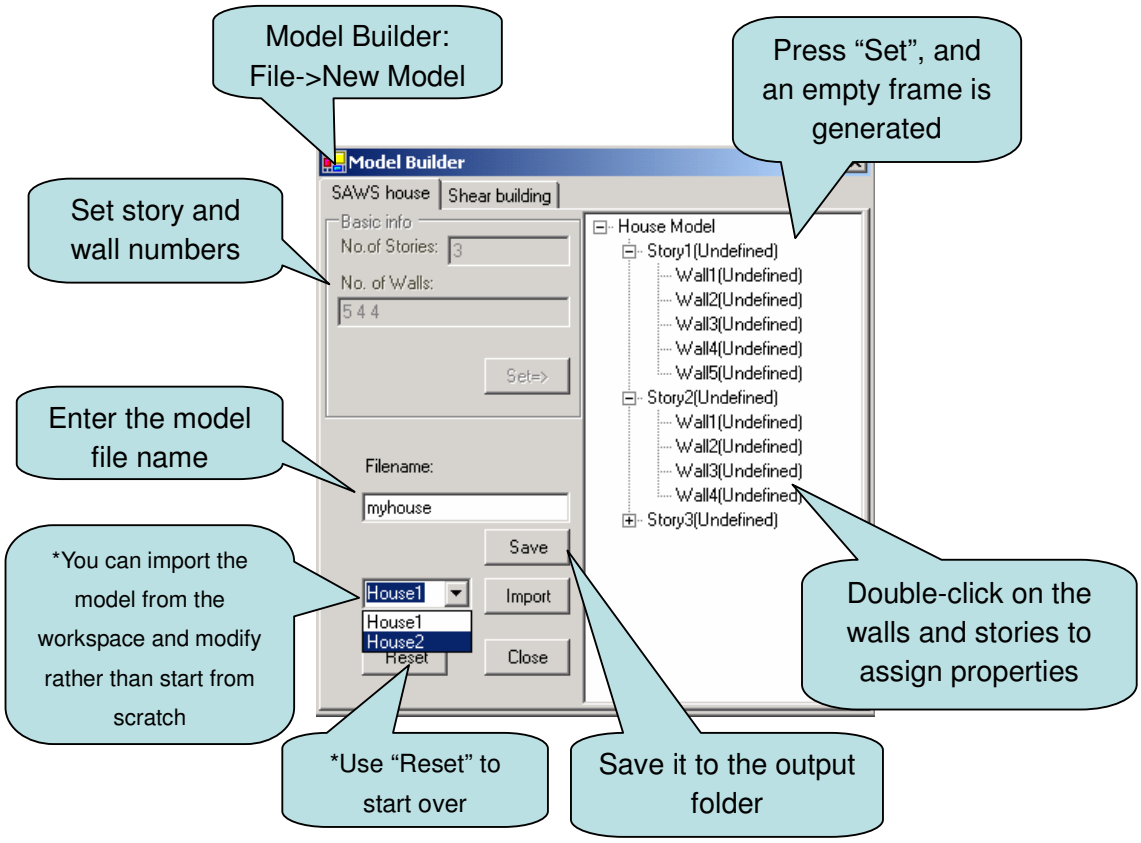


Figure 3. Model builder for house model

A similar procedure can be used in building a shear building model.

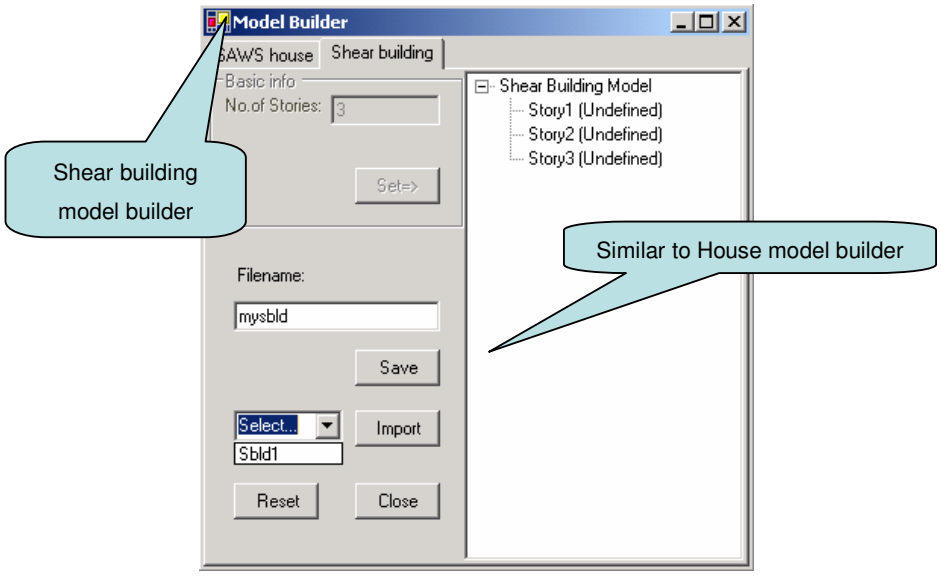


Figure 4. Model builder for shear building

The property window for a wall element is where you may enter the input wall direction, location, length, height, concentrated mass on top of the wall, wall model type (please refer to the Appendix for detailed information on the nonlinear models), and the hysteresis parameters associated with the model type. Other options for hold-down configuration are under development and will be included in SAPWood Version 2.0. Once the user clicks on "Update", the information for the wall will be stored in the model (it can be modified later by double clicking on the wall at any time). There are four model (hysteresis) types available in SAPWood currently: Linear, Bilinear, SAWS ten-parameter model, and a 16 parameter evolutionary parameter hysteretic (EPHM) model. Note that these hysteresis models can be mixed within a single house model.

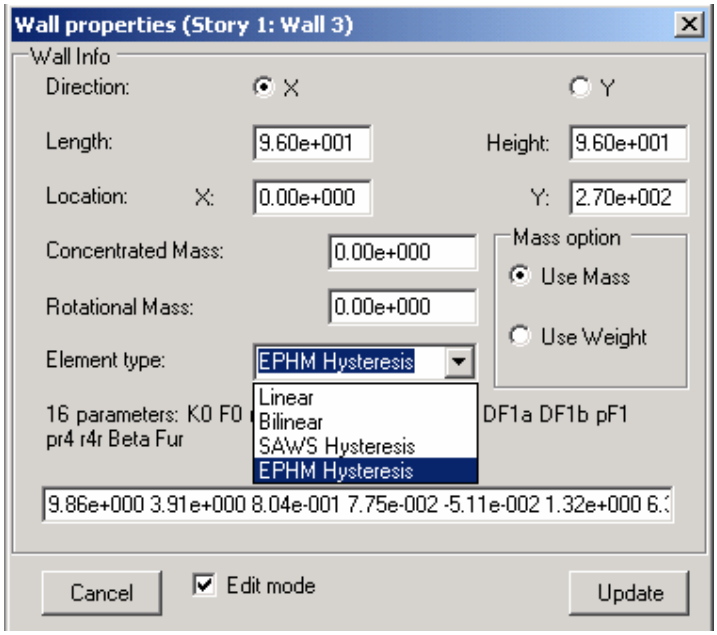


Figure 5. Wall properties window

The property window for a particular story level (in the simplified shear building model, the story element is equivalent to a wall element, that is, a non-linear spring) is the place where the shape of the floor diaphragm, distributed mass, and concentrated mass on the floor can be assigned. The nodes of the floor must be input in a counter-clockwise fashion. The user can plot the illustrative floor shape, locations of the concentrated mass, and wall locations (if wall information has been entered) to check the input by clicking "Plot and Calculate". The geometric properties and total distributed mass on the floor will be listed. The plot is not meant to be detailed, but rather to provide a ballpark summary for the user to identify gross problems with the input.

The user can choose between Mass and Weight as she/he inputs the masses. Mass units means the actual physical unit for mass (lb*sec^2/in for U.S. unit system and kg for S.I. system), and the Weight units are defined as lb in the U.S. system and Newton for the S.I. system.

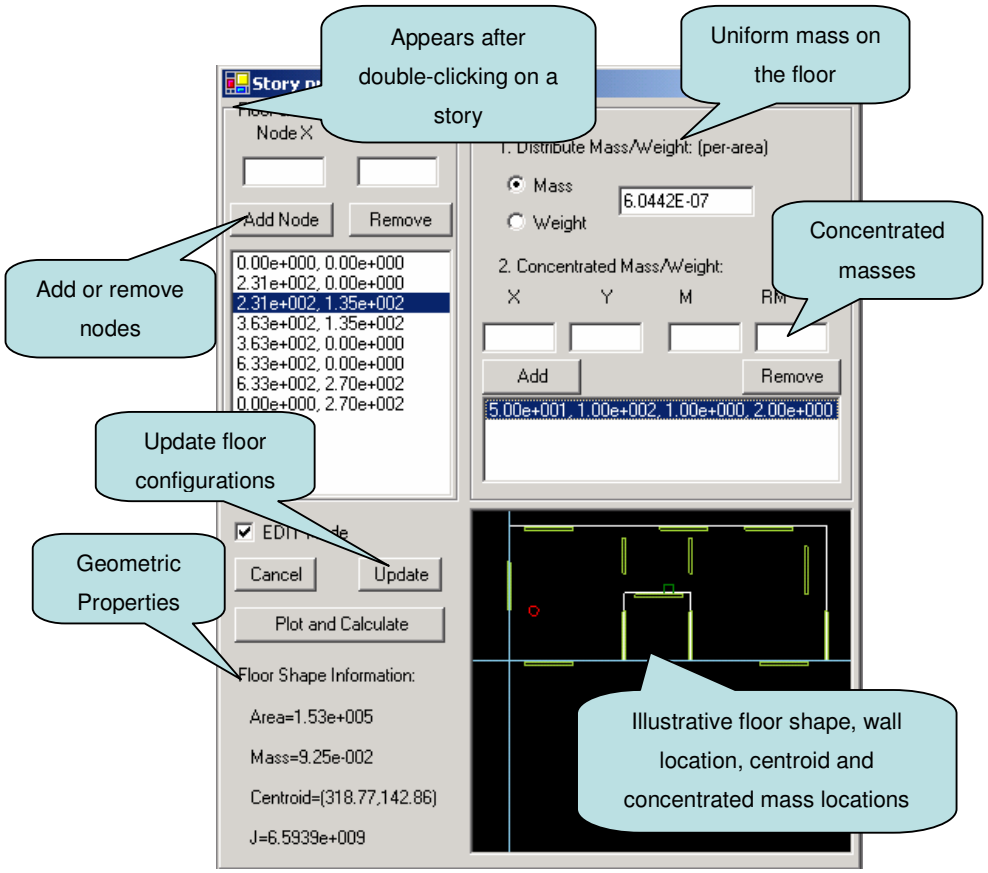


Figure 6. Story properties window

Tips:

- All of the model files are editable text files. Sometimes it may be easier to use a text file editor to create and modify the model files. Please refer to the input file format in the Appendix.
- All the locations (for walls, nodes, and masses) input into the model must be based on a single global coordinate system arbitrarily assigned by the user. It is usually easier (but not mandatory) to keep the entire plan view of the building in the first quadrant as shown in Figure 6.
- Both the biaxial model and the shear building model in SAPWood can be constructed with a mixture of different nonlinear/linear models.
- The nodes for a floor must result in a realistic geometric shape.
- If there is no distributed mass or concentrated mass on the floor, the user can input 0 for the distributed mass value and leave the concentrated mass blank.
- If the user does not want to assign any mass to individual walls, the mass and rotational mass of the wall can all be zero.

- Although the user is allowed to input and view the mass in both physical mass units and weight units, the data saved to the model file will always be in physical mass units. Be aware of this when you want to write a model file by hand using text file editors.
- The "Import" option in the model builder allows the user to use a model currently in the workspace as a startup frame for the new model. This can be very useful if one need to build a series of model with only a few differences in part of the elements, i.e. checking various designs of the same floor plan. It can also be used to transfer a SAWS input file to a SAPWood house model file. To import a SAWS *.dat input file, simply import the model and save it under a new name – SAPWood will recognize the file.

2.3 Single Earthquake Excitation

Purpose: To perform time domain analysis of a house or shear building model with an earthquake acceleration time series record and view/output the results of the analysis. This module also provides the modal information of the structure through fundamental period and mode. Higher order modes can also be obtained by performing Eigen value analysis on the dynamic matrix provided.

The use of this analysis tool is illustrated in Figure 7 in 6 steps:

Step 1: Load the earthquake record. The only earthquake file format SAPWood can use currently is the *.eq2 file, which is a text file with time T, Ax (acceleration in X), Ay (acceleration in Y) in each row, separated by a space. An example of the eq2 file can be found with the distributed software package. The units in the eq2 file MUST be in gravity (g), for example 0.6g is 60% of gravity. The program will multiply the eq2 data by gravity (depending on the choice of unit system) before analysis. This configuration allows the use of same input files for both the U.S. and S.I. system.

Step 2: Choose one structure from the workspace. This requires the user to load the model into workspace before loading the analysis window. Otherwise the model will not appear in the list of choice in the analysis.

Step 3: Input analysis parameters. Initially, the user can choose to scale the earthquake in both directions or use the un-scaled record by leaving the option unchecked. Since the analysis is by default bi-axial, you will need to put 0 in one direction if you want to do a uni-axial analysis. You will also want to choose an appropriate time-step value which will be used as the integration time increment in the analysis. If the time step is bigger than the time interval in the earthquake record file (can be read from the earthquake Info label after the earthquake is loaded), the record time interval will be used in the nonlinear time domain analysis. It is recommended to keep the time step below 0.005 sec if there are non-linear elements in the structural model. One way to examine if the time step is small enough is to look at the hysteresis plot after the analysis. If the corners of the resulting hysteresis are angular (with the obvious exception of the bilinear hysteresis), you may want to decrease the time step. The damping ratio here is a simplified way to apply damping based on the Mass Matrix and Initial Stiffness Matrix of the system (Please refer to the FAQ for details). The "Switch X-Y" box is an option to apply the earthquake record in X direction on to the Y direction of the model, similar for Y axis. Finally, the "time interval for maximum average acceleration" is a time interval used in the analysis to calculate the maximum average acceleration of each floor. Because numerical integration may result in irregular "spikes" in the acceleration time history, which does not reflect the realistic acceleration experienced by the floor, using the average acceleration in a given short time interval will help reduce the influence of numerical fluctuation. If the model loaded has the Friction Pendulum Base-isolation System built in, the "Enable FPS" check box will appear and the user can choose to enable or disable the FPS layer. "Include gravity" check box only applies to triaxial model and will not affect biaxial model or shear building model results.

Step 4: Press the analyze button and you will see the progress bar move from left to right. A typical 2 story house that contains about 20 wall elements under a 40 sec earthquake will take about 10-15 sec to analyze on an up-to-date PC. After the analysis is complete, the "Accumulate" button will be enabled, allowing the user to run another earthquake record (different records can be used by loading a new record) with the already damaged structure. This is a unique feature for SAPWood due to the use of the hysteretic model with damage-accumulation ability. If the "write elements" box is chosen, the hysteretic response for every walls and hold-downs in the model will be recorded, which will make the analysis slow down as time progresses because there will be less available memory. However, if the user is only interested in maximum values or just the global Dof responses, uncheck the "write elements" box will greatly accelerate the analysis process (this can be very important when a large number of simulations were conducted), the only information missing in the result will be the hysteretic loops for each wall element.

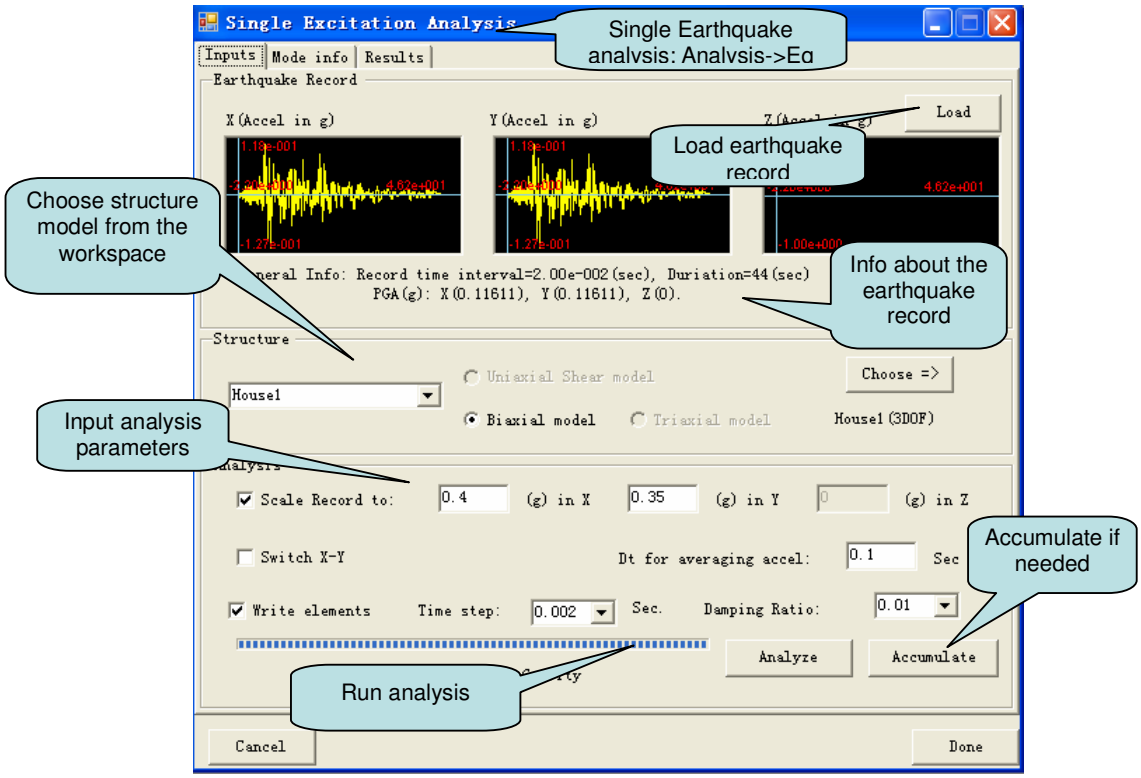


Figure 7. Single earthquake excitation analysis inputs

Step 5: Go into the "Mode info" tab where the fundamental period/mode, Mass matrix, stiffness matrix, and dynamic matrix can be obtained by clicking the "Calculate" button. The user can choose between "Initial value" and "Current value". "Initial value" is the modal analysis results from the initial stage (new structure). "Current value" indicates the modal information after the most recent earthquake excitation and is calculated based on the damaged structure. Typically, the damaged structure should have a longer period. An unrealistic long natural period or "NAN" result usually indicates the instability (failure) of the model.

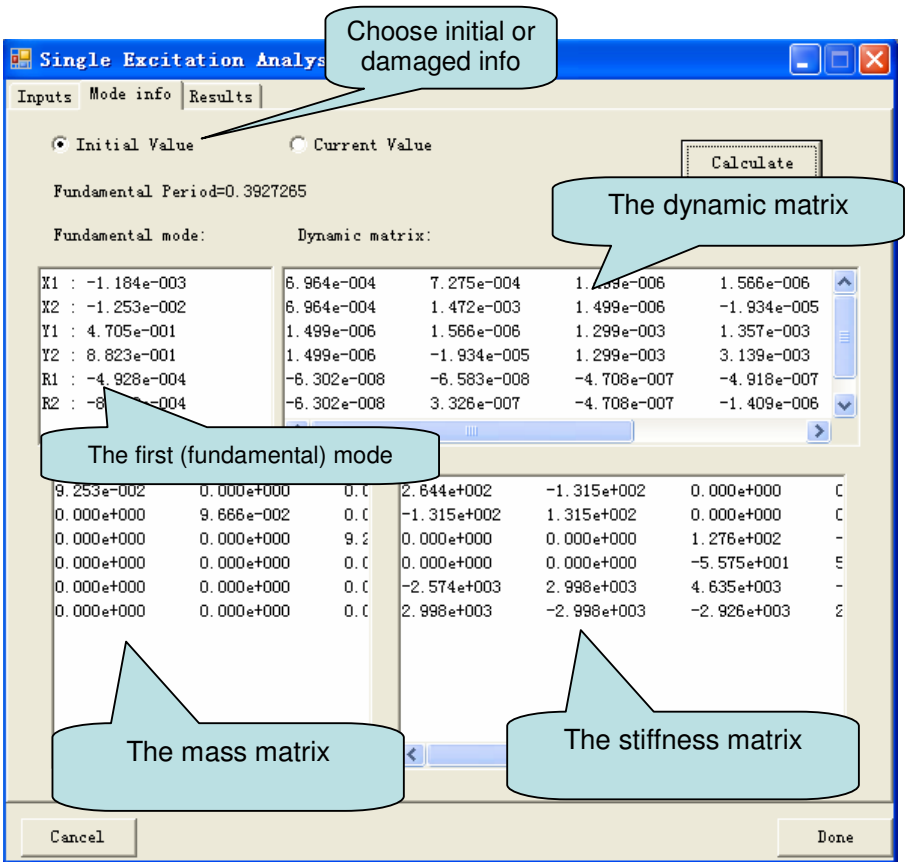


Figure 8. Modal analysis results

Step 6: After the analysis, the user can easily get a general idea of how the structure performed by looking at the results tab in the analysis tool. The user can also view the time history plot of any global DOF (including restoring forces and moments) and the hysteretic response of any wall, hold-down, or FPS element (if "write elements" box is checked during the analysis). The selected wall will be highlighted as the user browses among the walls. Then, the file output option leaves it to the user to decide which data should be saved as text files. General Info (*.gen) will always be output and it contains the maximum global responses. Global results (*.Glo) includes the time history results of global DOF's (force will not be included unless "Include force" is checked). The wall hysteresis (*.hys) contains the data of the hysteresis plots for every single wall.

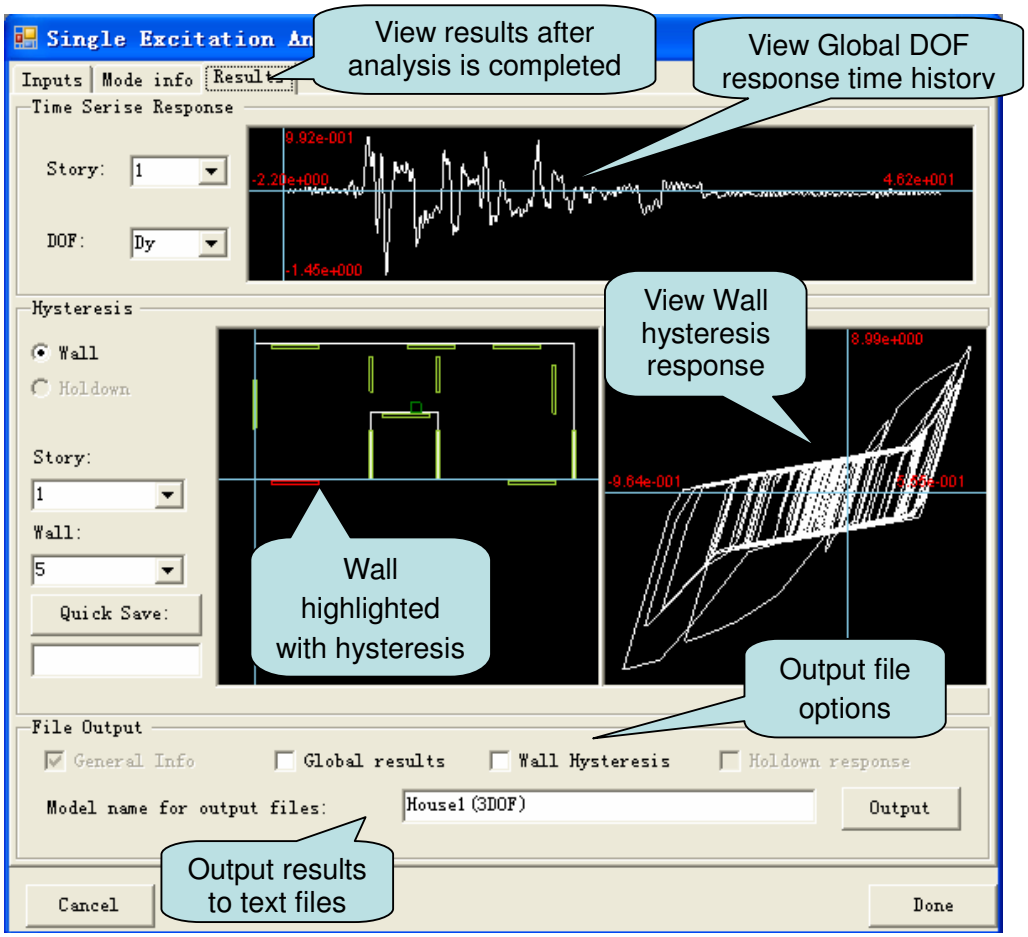


Figure 9. Single earthquake excitation results

Tips:

- Accumulation analysis can be performed to model successive earthquakes or repetitive excitation from a shake table. The results can be checked during the accumulation to observe the propagation of damage within the structure.
- It is easy to see if the elements behaved as expected by looking at the hysteresis plots. If the hysteresis plots are not satisfactory or the convergence is poor, one can decrease the timestep value to make sure that the hysteresis shape looks more appropriate.
- You may not need to output all results for every analysis; for example if you are using a trial and error approach to achieve a certain drift, you may only want to see the maximum response in the *.gen file.
- You must close the analysis window to load new models into the workspace. So load every model you need before opening the analysis window. This is true for other analysis windows also.
- If there is an unexpected error during analysis and the time-step is small, check the model first.

- If there is an unexpected error during plotting, check if your model has walls in only one direction (which is unrealistic and will cause a singular stiffness matrix). If so, the restoring force in the other direction will always be 0 and result in a plotting. The rest of the results are still good, plotting the force in that direction should be avoided under such a situation.

- If the results seem to be too large, it either indicates the structure is not strong enough or the time-step is too large. Also, be sure to check that you not entering weight values as mass.

- The Switch X-Y option switches the input AFTER the records are scaled. So the scaling is based only on the X and Y direction of the earthquake record, NOT the X and Y axis of the model.

2.4 Incremental Dynamic Analysis (IDA)

Purpose: To perform Incremental Dynamic Analysis on a single structure using a single earthquake record. View and output the analysis results (IDA curves) after analysis.

The use of this analysis tool is illustrated in Figure 10 in 5 steps:

Step 1: Load the earthquake record.

Step 2: Choose the structure to be analyzed from the workspace.

Step 3: Input the list of earthquake intensities (PGA or Sa) for each component of the ground motion. If "Use original ratio" box is checked, only input intensities for X component and other components will be scaled so that the intensity ratio between the components will be kept as the original earthquake record.

Step 4: Set the analysis parameters and click on the analysis button. The parameters are similar to a single earthquake excitation case, except the analysis will take significantly longer depending on the number of values in the PGA or Sa vector. Un-check "write elements" box will greatly help with the speed of analysis.

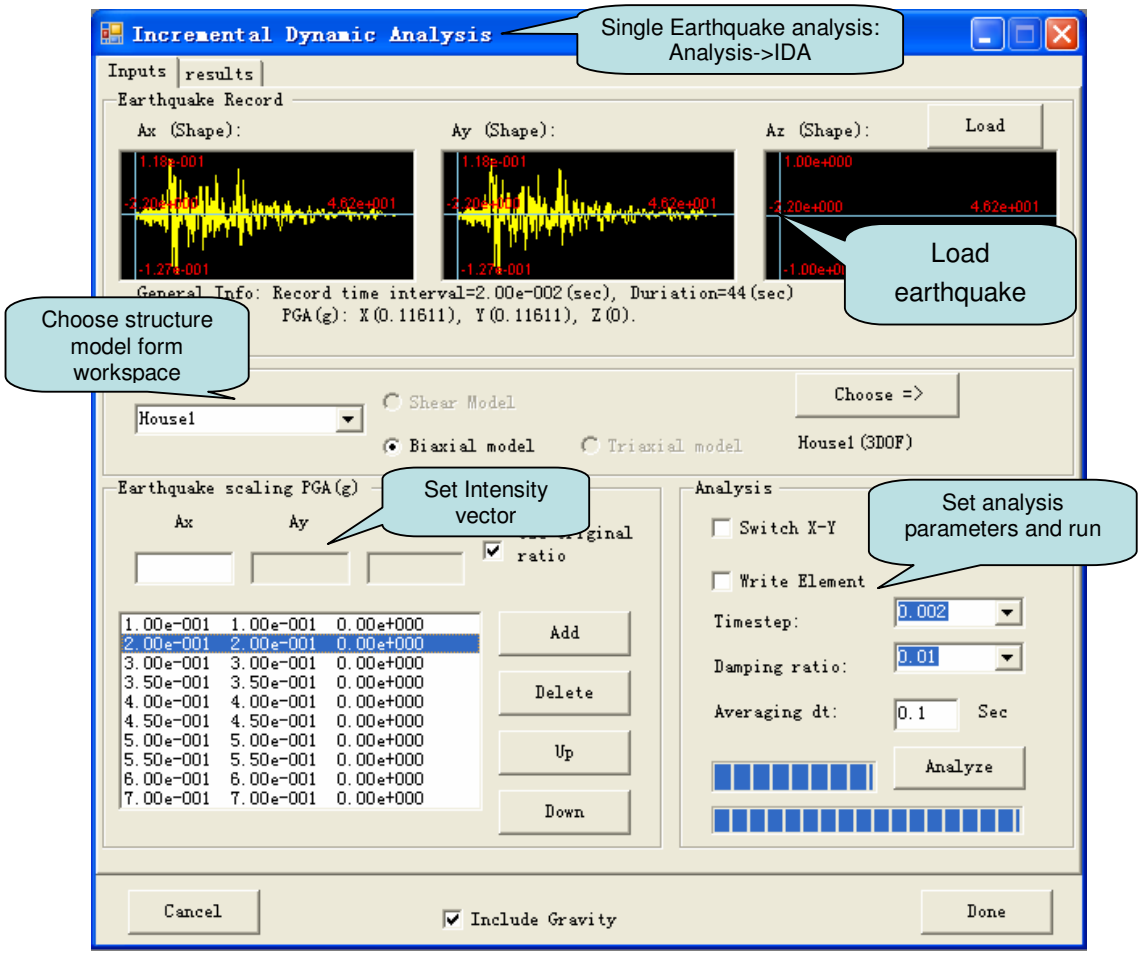


Figure 10. Incremental dynamic analysis inputs

Step 5: From the results tab, the user can plot the IDA of any global DOF or local (wall and hold-down) response. The numerical values of the plotted curve will also be listed in the table at the same time. Finally, one can output all IDA results to a text file (*.IDA) using the output option.

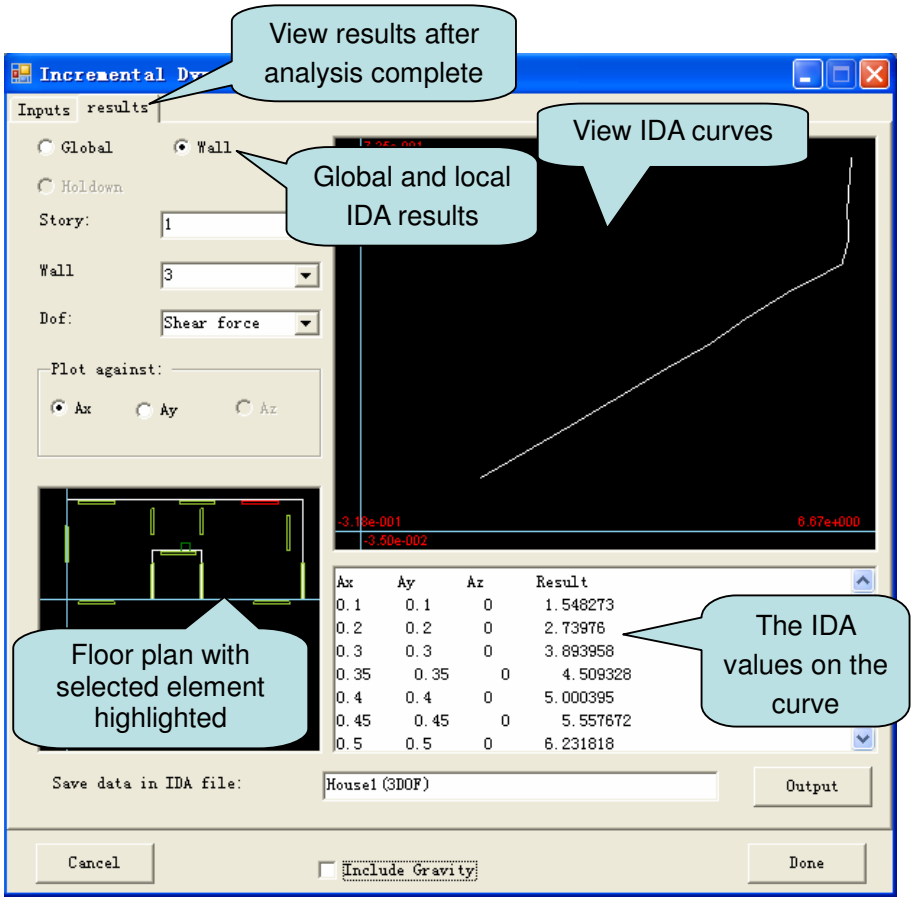


Figure 11. Incremental dynamic analysis results

Tips

- It is a good idea to run a few single analyses first in order to determine the maximum earthquake intensity you would like to use in the IDA if you're unfamiliar with the performance of the model.

- The plots generated in SAPWood are intended for illustrative purposes. If you need a plot of publishable quality, the plot should be generated using commercial software such as Excel or MATLAB.

2.5 SDOF System Identification

Purpose: To obtain the cyclic/monotonic load-displacement properties of a house system for a single global DOF by forcing it through a predefined displacement protocol. This option is only valid for biaxial models.

This tool provides a way to simplify the effects of the entire lateral force resisting system in a floor or entire system to a single degree of freedom nonlinear spring. It can then be used in a simplified structural analysis or to determine the load resisting properties of the entire structure. The use of this analysis tool is

illustrated in Figure 12 in 4 steps:

Step 1: Load the displacement protocol file. SAPWood can only read a displacement protocol from displacement protocol files having the following extension (*.pro). This should be a text file that contains only one column of displacement values (one on each row). Examples of a protocol file may be found in the distributed package.

Step 2: Select the structure model. Only a biaxial house model can be chosen in this analysis option.

Step 3: Set the analysis parameters and run. The protocol can be scaled so that the maximum value is equal to the value input by the user. The user also needs to select the global DOF of interest. The sub-step here is again used to divide the original displacement increment length in the protocol file to obtain the analysis displacement increment.

Step 4: The hysteresis of the global DOF can be output to a text file for future use. The file will be in the output directory with the extension *.ehy.

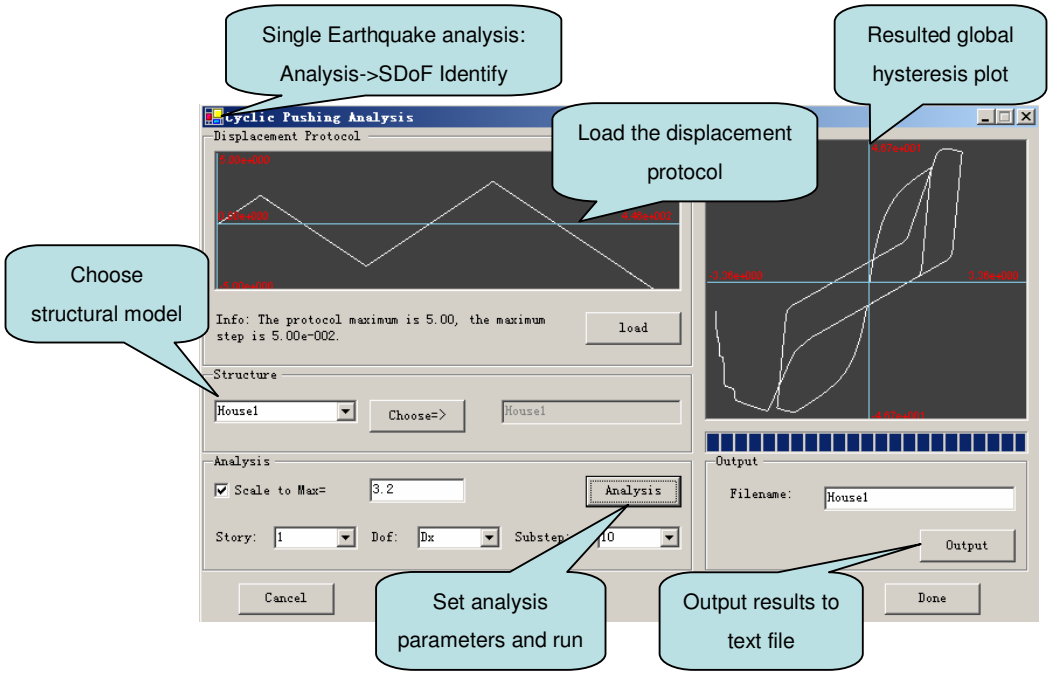


Figure 12. SDOF identification analysis

Tips

- The house model should have walls in both directions to eliminate unexpected numerical error and instability.
- The program might allow the user to push the structure to unrealistic (very large) displacement values. But the user should use their experience to determine the credibility of the results. For example, if one 'pushes' one story of a wood frame structure horizontally for 3 ft, the results up to a ft or maybe 1.5 ft are usable.
- Try increasing the sub-step value if the plot does not seem to be a reasonable looking hysteresis.
- The protocol file can be very "coarse" (containing only the key points), then simply use a large sub-step number to get a refined result.

2.6 Multi-Record IDA

Purpose: to provide an automatic tool to compare the dynamic properties of multiple structures under multiple earthquake ground motions and intensities. It can also be used as the tool to perform batch analysis.

The use of this analysis tool is illustrated in Figure 13 in 6 steps:
Step 1: Choose multiple models. The models do not need to be in the workspace. Only the same type of model (Biaxial or shear building) is allowed for analysis in any one Multi-IDA. The file name of the model will be added to the list after it is loaded.

Step 2: Select multiple earthquakes. The file name of the earthquakes will be added to the list after each one is loaded.

Step 3: Enter the PGA or Sa vector. This is very similar to the IDA module. However, if the user just wants to run a group of earthquakes without scaling, the "No scale" box can be checked to achieve that.

Step 4: Set the analysis parameters. This includes the model type option (so a single MIDA run can only include one type of model), time step, and damping (ratio to the critical damping). Note that the time step is used here instead of a sub-step since different earthquake records might have different time increments. The program will automatically assign different sub-steps to each earthquake so that the time increment is a constant across all earthquakes. Again, un-check "write elements" box will greatly improve the simulation speed, especially when the simulation amount is great.

Step 5: Choose output options. Checking on "Analysis log" will provide a *.log file recording the analysis inputs and parameters. "MDA file" will generate *.MDA file with all the maximum responses. By checking "Output gen" and "*.glo", the analysis will output all the *.Glo and *.gen files for every single analysis performed in the MIDA. This is equivalent to a typical batch run analysis.

Step 6: Run the analysis. Since the multi-case IDA may take significant time to run, the user should note that MDA files will be automatically generated only after all time domain analyses are completed. Having the program to output *.gen and *.glo for each analysis will be helpful in case of a possible system crash. If a project name is assigned by the user, these files will be saved under the assigned project name. Otherwise, the program will generate a file name based on the time and date of the analysis automatically.

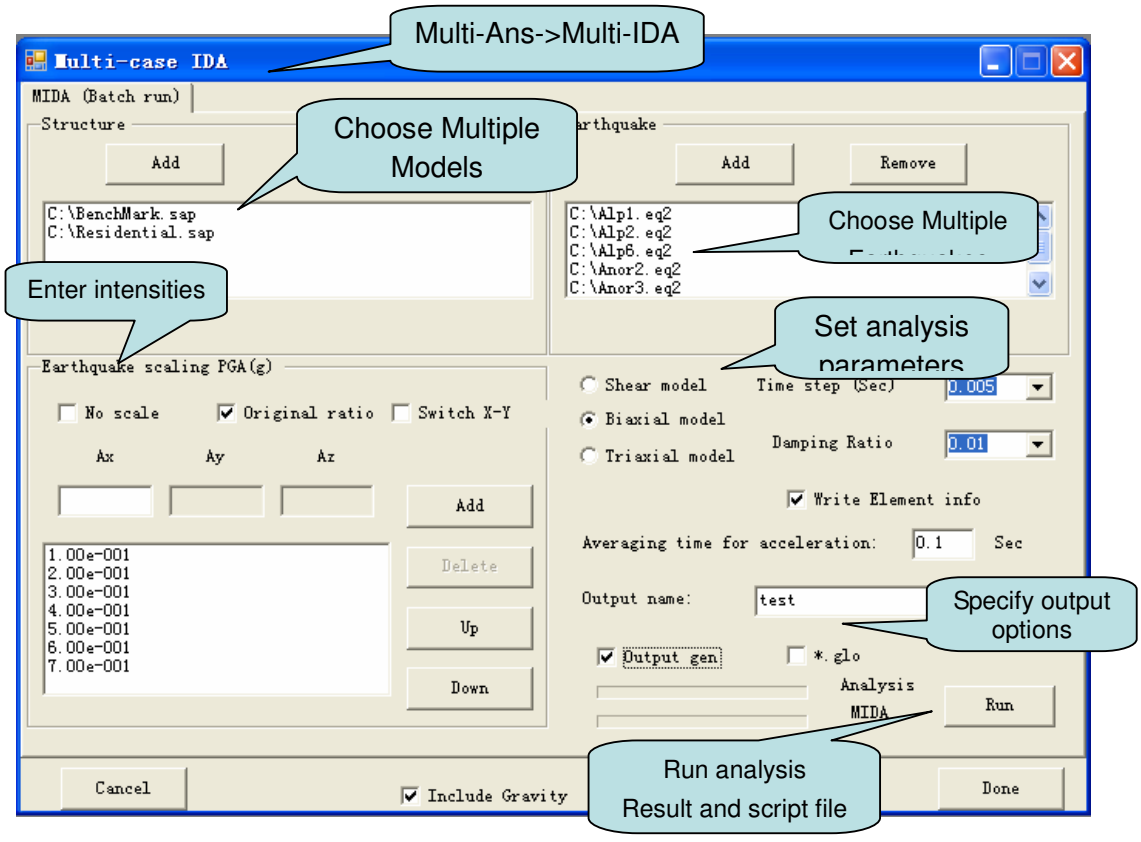


Figure 13. Multi-case IDA inputs

The user must use the output files from MIDA analysis to evaluate the analysis results.

Tips

- Multi-case IDA is often a very computer-intensive procedure. The program will not respond during the process, so one can just minimize the window and let it run. It is always a good idea to estimate the total time by multiplying the single analysis time by the number of earthquakes, structural models, and PGA/Sa points.

2.7 Manual Fitting tool

Purpose: To provide the user with a tool to determine nonlinear spring model parameters based on an experimental hysteresis or hysteresis result from SDOF Identification.

The use of this analysis tool is illustrated in Figure 14 in 5 steps:

Step 1: Load the hysteresis file. Any text file with hysteresis data organized in 2 columns (first column is drift, second is force) can be loaded into SAPWood. The hysteresis will appear in red after it is loaded.

Step 2: Choose the model type to be fitted (Linear, Bilinear, ten-parameter model used in SAWS, and EPHM). Once a model type is chosen, the program will automatically generate a set of trial parameters in the parameter list. Then the user can start with these parameters, modifying them to obtain a better fit.

Step 3: Try different sets of parameters. Every time the parameters are altered, click on "fit" to see the results of fitting (the white line is the model hysteresis).

Step 4: Alter parameters till satisfied. The method to alter parameters for a particular model differs depending on the model itself. Please refer to the model description in the appendix.

Step 5: The user can output the plotted data to a text file so that it can be used later in other programs. Both the original data and the model response will be outputted.

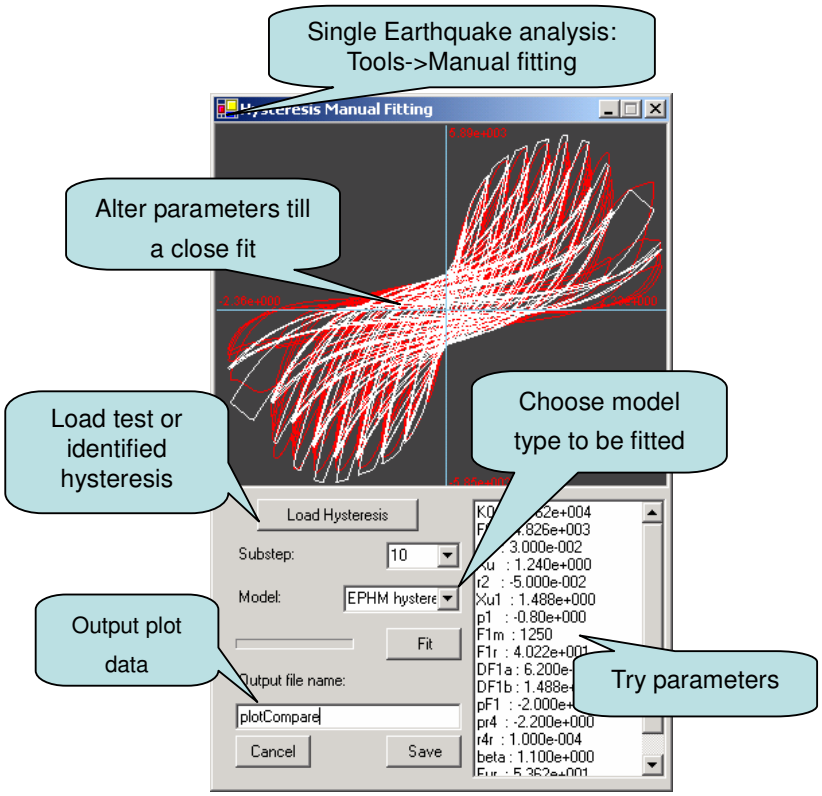


Figure 14. Manual fitting tool

2.8 SAPWood-Nail Pattern

Purpose: To provide the user with a tool to build and analyze shearwall assemblies using non-linear hysteretic nail/connector elements. It is similar to the CASHEW program developed in CUREE-Caltech

project. This module currently includes two parts: the model building and analysis tool. More convenient model building options will be added in the future.

2.8.1 NP Model Builder

Purpose: This is an interactive tool to build shear wall models with components (studs and panels) and connectors (nail, screw, etc.). The model built in this tool can be loaded into the SAPWood-NP Analysis module to perform monotonic or cyclic loading analysis. The NP model has 3 degree-of-freedom on every component (stud and panel). These components undergo rigid body displacement with the restraint of the nails connecting them together.

It is highly recommended that the user use a text editor to construction NP model files based on the format explained in the "Model file format instruction.xls" file distributed with this package. For a complicated model, using the interactive model builder may be less efficient than directly create model files.

The use of this tool is illustrated in Figure 15 in 6 steps:

Step 1: Load the NP model builder.

Step 2: Enter the total number of studs and panels in your wall, and press "Set". A list of studs and panels with an automatic numbering sequence will appear in the list box below.

Step 3: Input the center location in global coordinates (It is up to user to establish the global coordinate at any location in the wall plane, but the X direction must be horizontal and Y direction must be vertical. This coordinate system, once established, will be used for both component location and nail line location input). It is recommended that the user always set the 1st stud (#1) as the top stud. Then the "pushing" DoF during the analysis will always be "1X". The user must also set at least 1 fixed component (usually the bottom stud) as boundary condition to fix the wall. Just think of the model as a wall being tested in a cyclic loading test, all the studs that will be bolted to the ground should be assigned as "fixed" in the model builder.

Step 4: After all the component information has been given, the user can input the information for nail lines. This part is similar to the CASHEW program, the user needs to specify the start and end points of the nail line, the nail spacing, nail parameters, and the components this nail line connect. NP builder offers input options of inputting nail numbers, direction, and spacing instead of start and end points (when you choose to input with start and end points, direction will always be "other", which allows you to use nail lines in any direction.). Nails in a SAPWood-NP model can be a mixture of different hysteretic model type. Linear and bilinear models can also be used. Nail lines can connect studs and panels as they do in CASHEW; they can also connect stud to stud so that framing nail behavior and hold-downs can be modeled (in this case, the user might use a nail line that contains just 1 or 2 framing nails).

Step 5: Apply the nail line after all the information has been entered, the nail line will be plotted in the plotting window below and added to the Nail schedule list box on the left. Check if there is any error in the input. If there is, the user can delete the nail line he just entered and re-enter the information again.

Step 6: After all nail lines have been entered, save the model under the file name of choice.

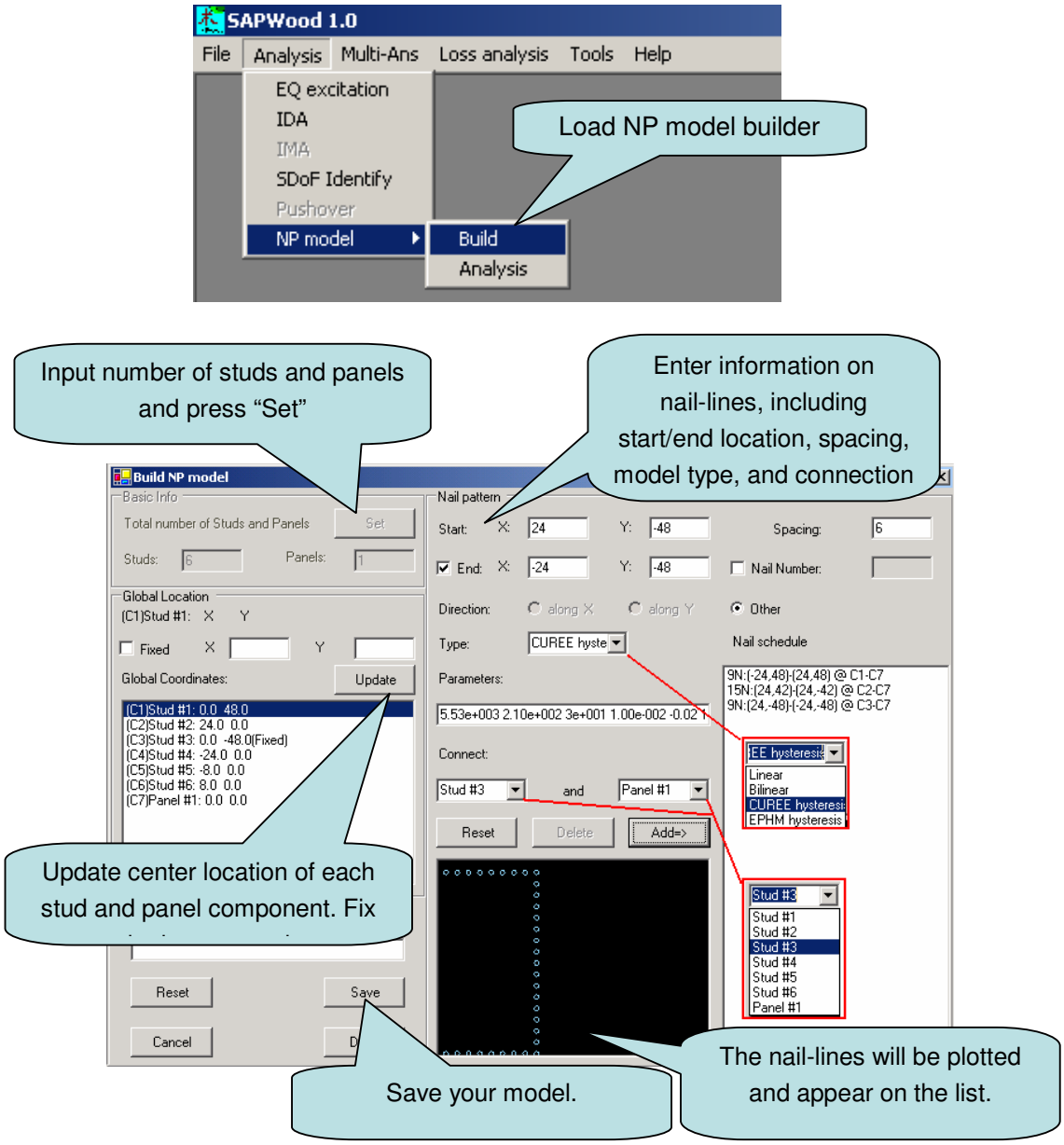


Figure 15. SAPWood-NP model builder tool

Tips

- Some users might find it easier to create the model file with a txt editor rather than using the model builder. Please refer to the NP model file format in the Appendix for file format information. The model file has an extension of *.NPS.
- The NP model does not allow 2 nails in the same location. If such a configuration is needed, the user can shift one of the nails with a very small amount of displacement (such as 0.01 inch) to avoid an error message.
- The unit system in the NP model is the same as the global models. Always be consistent with the unit system in which your nail parameters were constructed.

2.8.2 NP Analysis

Purpose: This is an interactive tool to perform monotonic or cyclic loading analysis on the shear wall models built from the NP model builder.

The use of this tool is illustrated in Figure 16 and 17 in 5 steps:

Step 1: Load the NP Analysis tool.

Step 2: Load the protocol file. This will be the displacement protocol the wall is to be displaced with. It can be either monotonic or cyclic.

Step 3: Load the model and the plot of nail lines will appear. If there is an error message, please check the model file.

Step 4: Set analysis parameters and analyze. The analysis parameters include: protocol scaling parameter which allows the user to scale the protocol to a maximum value, sub step parameter which divides the protocol increment into sub steps, the pushing DoF which allows the user to choose the DoF of the model on which the protocol is to be applied (typically the transient DoF in X at the top stud for a shear wall analysis), and the Check nail option that makes the program store the hysteresis loop for every nail during the analysis (can be used to detect the most damaged location in the loading process and/or nail failure). The analysis will be slowed down if the "Check nail" option is selected since there may be hundreds of nails to store in memory.

Step 5: Check results after the analysis is completed. The hysteresis loop for the wall at the Pushing DoF will be plotted immediately after the analysis. The user can save the hysteresis to a file for future use (such as fitting with the Manual fitting tool or loading into anther program for plotting purposes) or begin to fit the wall level parameters in the analysis window (the procedure is exactly the same as Manual fitting tool). If the Check nail option was selected before analysis, the user can plot the hysteresis loop for every nail in the X and Y direction to see if the connection has suffered degradation, if desired. In addition, the

deformed shape of the nail lines can be viewed by clicking on the "Max" button above the nail line plot. The "Initial" button is to bring the plot back to the un-deformed status for comparison.

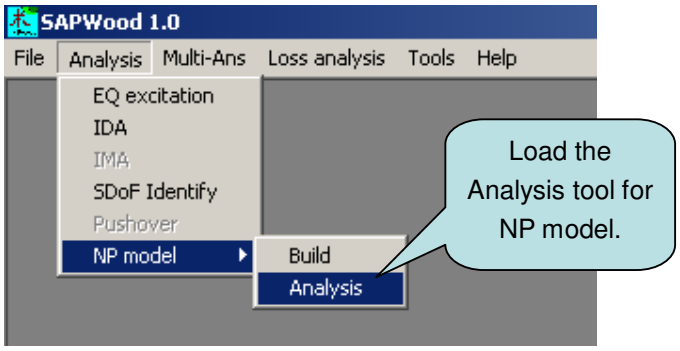


Figure 16. Load the SAPWood-NP Analysis tool

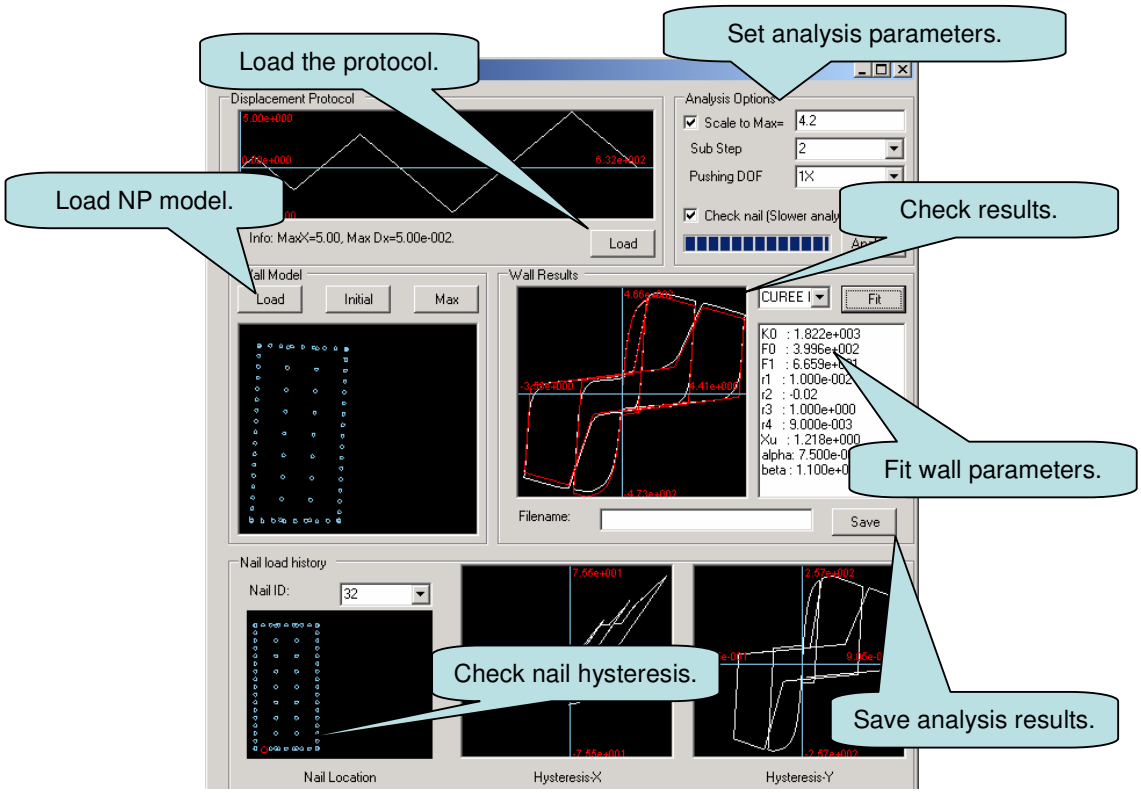


Figure 17. SAPWood-NP Analysis tool

Tips

- There might be situation that the NP analysis results do not look like a realistic hysteresis. This might be the result of excessive displacement. Keep in mind one can only push the wall within a limited range of displacement.
- If the analysis goes unstable very fast during loading, check to see if one or more components are not constrained enough to prevent rigid body motion.
- Changing the number of sub-steps may help convergence.

2.9 Wall parameter database

Purpose: The wall parameter database is a toolbox to generate shearwall and drywall parameters for different wall height, length, nail pattern, etc. It was developed from NP-analysis results for standard wall segments. So it should only be applied to solid wall segments.

Currently, the wall parameter database is developed under only the U.S. unit system. This database serves only as a reference in preliminary stage of analysis where the wall test data is not available and detailed NP analysis is not necessary. The model for drywall in the database is ten-parameter CUREE model. All shearwalls are modeled using EPHM.

The use of this tool is illustrated in Figure 18 in 3 steps:

Step 1: Load the wall parameter tool box, read the warning information.

Step 2: Load the wall parameter database file (a database example "Walldata.bas" file is attached).

Step 3: Specify wall configurations, input wall length (in inches), click on "Get parameters". The parameter for the wall will appear in the list on the right side.

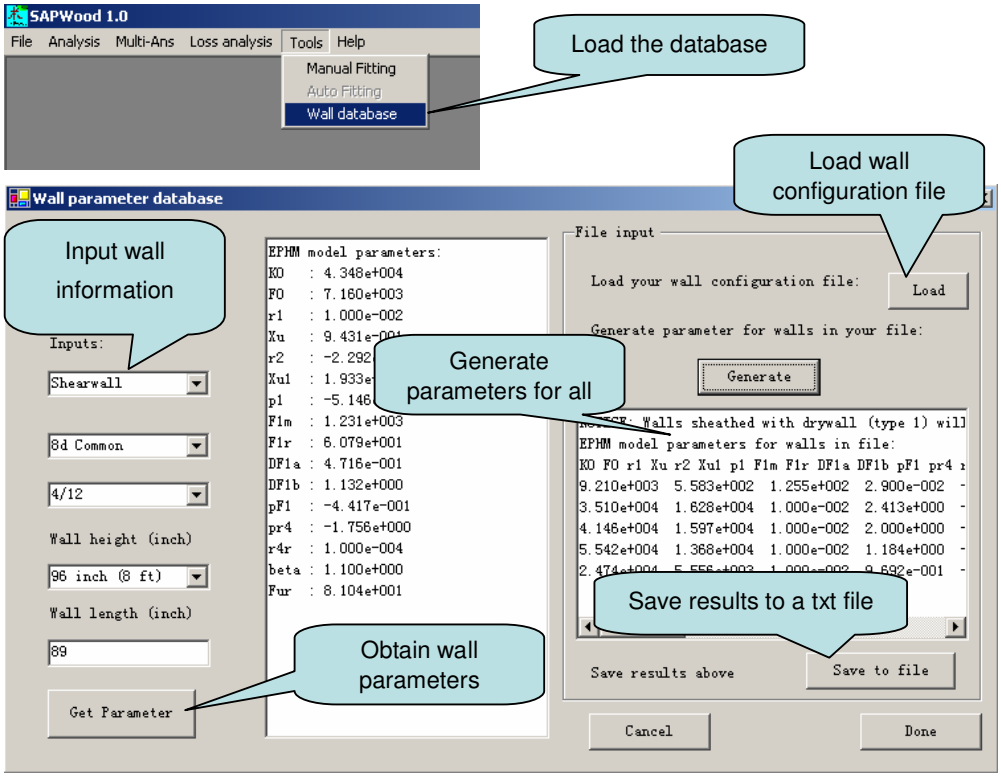


Figure 18. Wall parameter database toolbox

This procedure is good for a small number of walls. If there are a large number of wall configurations that need to be evaluated, the other option is to create an input file containing all wall information, generating wall parameters all at once. The format for the wall configuration file can be found in the Appendix: Input file formats.

Tips

- If the S.I. unit system is desired, one should use test data or SAP-NP model rather than convert the U.S. unit based parameters in the database, since there are dimensionless exponential decaying parameters which may be difficult to transform from one system to the other.

- The shearwall database should only be used in basic analysis to help gain a feel for the structural behavior. There is large variation in the hysteresis of shearwalls even with identical nominal configurations. This variation is not considered in this parameter database.

2.10 Loss analysis

Tools to simulate and predict economic loss in woodframe structures under earthquake hazard analysis are integrated into SAPWood based on the long term loss analysis framework developed by the author (2007) and the assembly based vulnerability framework proposed by Porter (2000). Two simulation procedures are included in the loss analysis module. Vulnerability based simulation first generates single earthquake

vulnerability model for the target structure through time domain analysis simulation; then the vulnerability result file is loaded to perform long term loss simulation. A Virtual Hazard procedure simulates the entire life-time of the structure with successive time domain analyses on a damage accumulation model. Details of both simulation procedures can be found in Pei (2007).

The Loss-analysis module released with SAPWood 2.0 has not been fully calibrated. The user should be aware that the results of the loss analysis depend on the user's own input information and do not represent the opinion of the authors.

In addition to the structural model, additional cost and hazard related information for the structure needs to be inputted for the analysis. In the current version of SAPWood, these inputs are organized in form of input files including an Element information file, Component fragility file, and Hazard environment file. The Element information file presents the damageable elements inside the structure and the way in which the damages are related to structural responses of the structural components. The Component fragility file presents the relationship between structural response and repair cost for each type of component. The Hazard environment file indicates the earthquake record suite, earthquake occurrence information, and hazard curve data. These files are used in both simulation procedures in SAPWood. The detailed information on the format of these files can be found in the Appendix: Input file formats.

The use of input files maximizes the ability of researchers to customize the analysis. However, it might make it inconvenient for practitioners to use this module. This situation will be addressed in a later release of SAPWood. Currently, the loss analysis module is designed primarily for research purposes rather than practical applications.

2.10.1 Vulnerability based loss simulation

Purpose: Perform vulnerability analysis for a single woodframe building to obtain vulnerability model (distribution of single earthquake loss as function of seismic intensity). Then this model can be used to perform long term loss estimation by generating long term loss samples given exposure period.

Assuming the input files have been properly prepared, the vulnerability based loss simulation can be performed following steps, as shown in Figure 19:

Step 1: Load the vulnerability based simulation tool.

Step 2: Enter a seed value for the simulation. By setting the same seed value (must be a positive value) for different analyses, it will control the random number generator to yield the exact same random array. In this way, sensitivity analysis and comparison can be performed excluding the effect of the random number generator uncertainty. If no number is entered, the generator will not be controlled and generated by the computer.

Step 3: Select the reduction level of strength and stiffness of the structural model. By setting the maximum and minimum reduction factor, the strength and stiffness parameter of the structural elements will be randomly generated in between. The deduction for every component will not be the same, but follows uniform distribution between the boundaries. This input is designed to represent construction quality and

uncertainty.

Step 4: Load the structural model, which must be the biaxial system model. Then specify the number of random model samples to be generated. For each time domain analysis of a single model sample, specify the number of loss samples to be generated.

Step 5: Load the Element information file. This file provides the information about all elements that may cause loss during an earthquake, with each one connected to the structural response (drift, displacement, or acceleration, etc.) of a structural component. It also indicates the element types and the actual replacement cost for each component. Adding all the component replacement costs together, the total loss is the collapse loss which is the maximum loss that could occur during an earthquake.

Step 6: Assign damage fragility to each damageable element type indicated in the element information file. Damage fragility distributions in SAPWood are constructed using a Bayesian framework. The user needs to specify in the Component fragility files the boundaries for the uniform priors of response and normalized cost given damage level, then provide data to update the priors.

Step 7: The Load Hazard environment file, in which the location of earthquake records are specified. The information on earthquake occurrence rate, hazard curve for the building site, intensity indicator (PGA or Sa) are also indicated.

Step 8: Enter the intensity grid for vulnerability model results. The input and concept are similar to the IDA intensity array. But the intensity grid here should start from the intensity which is believed to cause no loss (very small) to the intensity which will almost surely collapse the structure (very large) in order to obtain a complete understanding of the vulnerability behavior.

Step 9: Enter the output name (SEC extension will automatically be added indicating Single Earthquake Cost), click on "Vulnerability Analysis". The analysis may be very time consuming if a large amount of samples are to be simulated. The total number of time domain analyses can be calculated by multiplying the number of model samples, the number of earthquake records, and the number of intensity values together.

Step 10: After the SEC file is generated, load the file into the tool box and input the long term loss simulation parameters. The intensity limits indicate the minimum intensity that may induce loss and the maximum intensity under which the structure will definitely collapse. Ideally, the values should be the maximum and minimum intensity in the vulnerability analysis. The Exposure period should be in years since the occurrence of earthquake is measured yearly in SAPWood.

Step 11: Specify output name and the click on "Longterm". Desired number of long term loss samples will be generated and written to a *.LOS file which can be opened with a text editor.

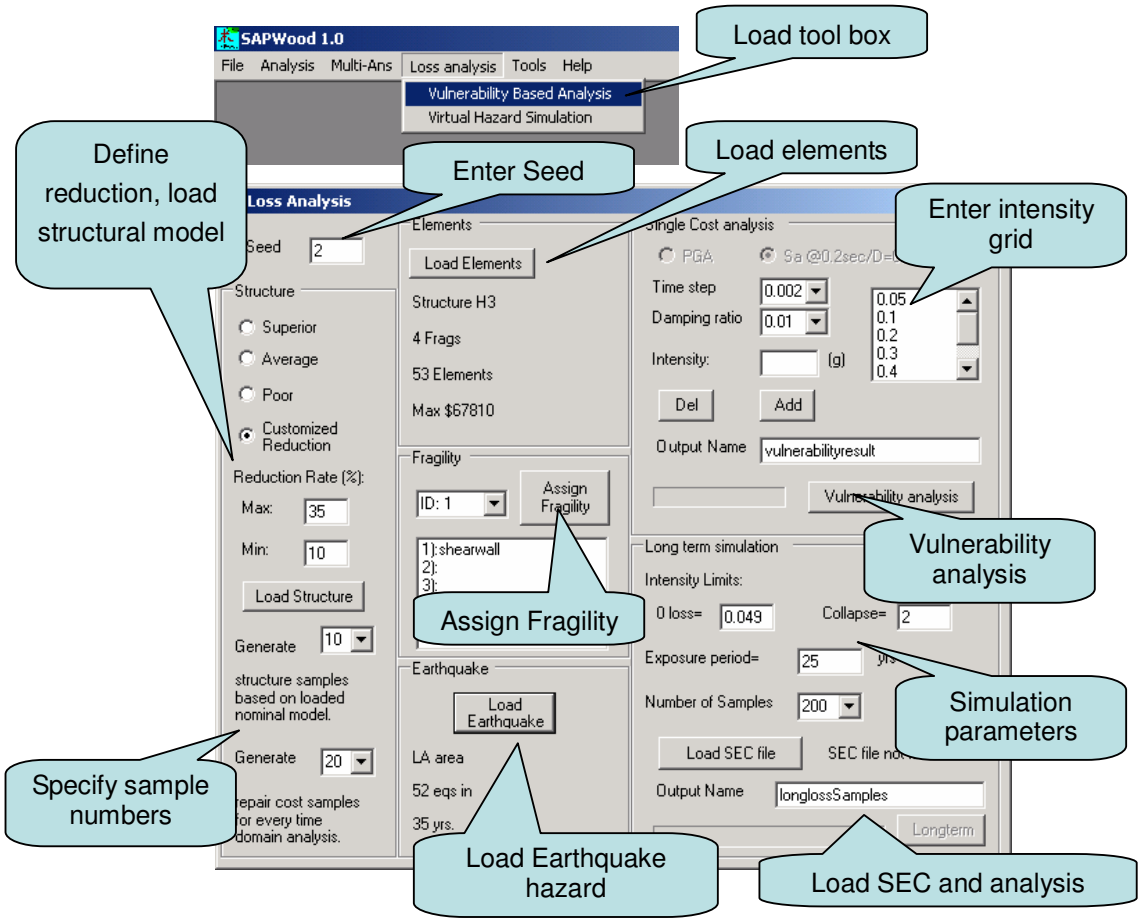


Figure 19. Vulnerability based loss simulation tool

Tips

- The theory behind the loss simulation procedure was described in detail in Pei (2007). A paper on loss estimation is under review at the journal of Structural Safety.
- SAPWood does not provide any post-processing model or procedures for the long term loss data generated. The conclusions and inference of the simulated sample may be obtained through the statistical methods or models based on a user's specific need.
- The user can even develop his/her own simulation scheme for long term loss simulation. The vulnerability analysis result *.SEC file can be used as an intermediate result for customized loss analysis. It is independent of the long term simulation scheme used in SAPWood.
- The loading of component fragility file will take some time because of the numerical integration needed for Bayesian updating. The program might appear to freeze for a short time but please be patient, it will finish.

2.10.2 Virtual Hazard simulation

Different from vulnerability based simulation; Virtual Hazard simulation repeats the loading history during the exposure period with a model that incorporates damage accumulation, performing time domain analysis on the structure with "cumulative damage" upon every earthquake occurrence. Many realistic situations can be studied using this simulation procedure, such as aftershocks, repair strategies, and non-visual damage, etc.

The basic input information for this type of simulation is identical to the vulnerability based method. The same element information file, component fragility file, and hazard environment file can be used. Please see the discussion for vulnerability based simulation for the details on related steps. The simulation can be performed with following steps:

Step 1: Load the simulation tool box.

Step 2: Specify the seed for random number generator.

Step 3: Load structural model, specify reduction factors.

Step 4: Load element information, assign component fragilities.

Step 5: Load earthquake hazard environment information.

Step 6: Specify repair strategy. Currently, the only option is to choose a damage level as the lower limit to apply repair. More options might be added in a later version of SAPWood.

Step 7: Specify aftershock generating parameter. These parameters control the minimum earthquake intensity that will be used as an aftershock, the probability of aftershock, and the range of the aftershock intensity as a function of the main earthquake.

Step 8: Specify simulation parameters. Most of the parameters are similar to the vulnerability based simulation. The only exception is the damage log option where the user can obtain a time-line of earthquake loading and economic loss history for every exposure simulation. So the maximum number of damage logs is the number of samples to be generated. However, one can choose to only generate a log for part of the simulation by specifying a smaller number or even specify no damage log at all. It is not a required option for the analysis.

Step 9: Click on "Simulate". The Virtual Hazard simulation is very computationally intensive. The analysis may take considerable amount of time depending on the number of samples and earthquake occurrence rate. An estimation of running time can be calculated by multiplying the number of sample, exposure period, average number of earthquake per-year, and duration for a single earthquake analysis together.

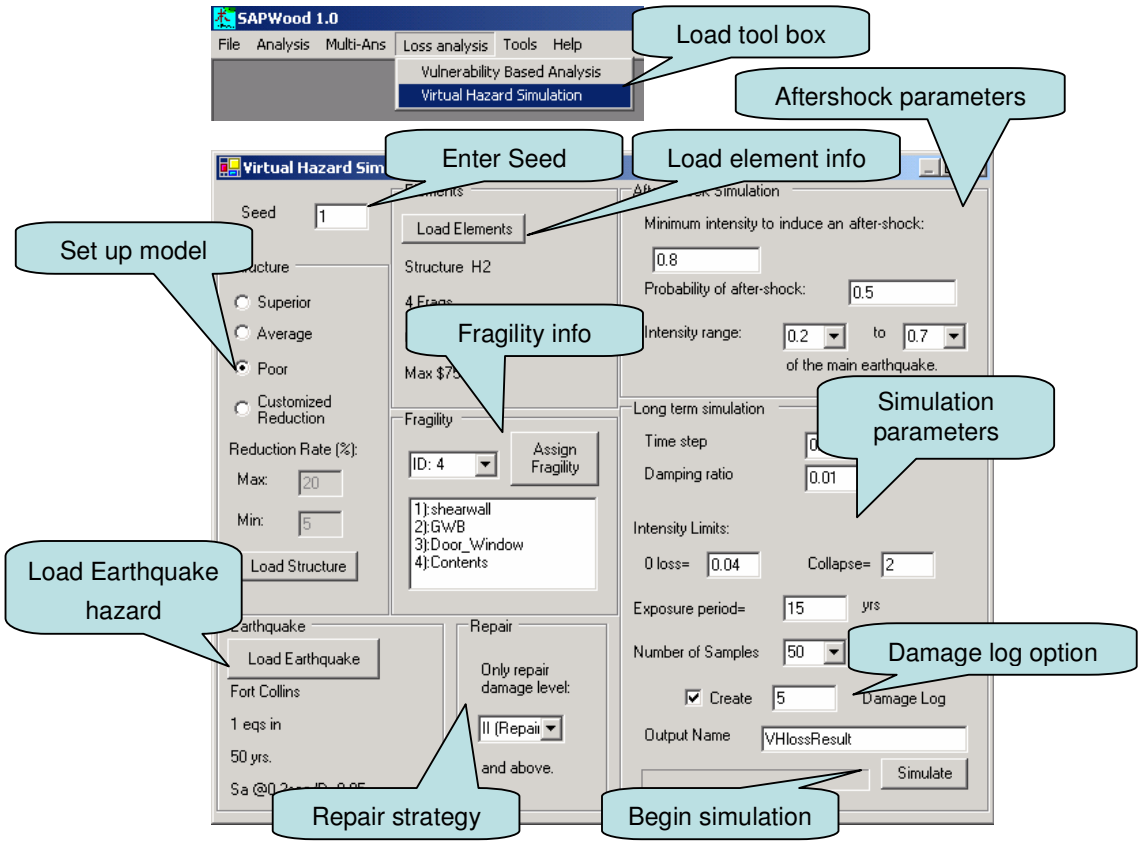


Figure 20. Virtual hazard simulation tool

Tips

- The Virtual Hazard simulation cannot provide vulnerability results as intermediate output.

3 Trouble shooting

3.1 Can't launch the program

This situation occurs in the Windows systems when the .net framework is not available, such as windows98 or early versions of windows 2000, NT, and XP. SAPWood was developed using VB.net and requires the .net frame work to run. The .net framework will enable future versions of SAPWood to be run over the web with file sharing, etc.

To solve this problem, the user should go to Microsoft webpage:
http://www.microsoft.com/downloads/details.aspx?FamilyID=0856eacb-4362-4b0d-8edd-aab15c5e04f5&DisplayLang=en
and download "Microsoft .NET Framework Version 2.0 Redistributable Package" and install this package on the system. This package can also be found by searching in (www.microsoft.com/downloads) for

"dotnetfx.exe". This file is about 22.4MB and is free.

If you still have problem, please contact the authors: Shiling Pei (slpei@lamar.colostate.edu) or John W. van de Lindt (jwv@engr.colostate.edu).

3.2 Fail to load model file

If the filename is correct and the file will not load, please check the data in the file. The *.sap and *.sbd file must all use spaces as separators. There should not be any blank line in these files. (This is easy to achieve if you use the model builder to build the file rather than write the file in a text editor yourself)

The *.dat file (SAWS file) can use ",", space, tab, or a combination of these as separators. The *.dat file can use "!" as a comment indicator. The empty line between data is allowed in this type of file.

It is recommended to use the model builder to convert your *.dat file into a *.sap file. Just load the *.dat file into the workspace, open the model builder, import the model, and save it under another name.

3.3 Unexpected termination during analysis

This might be caused by model instability (e.g., excitation in Y direction but no wall in Y direction) or lack of convergence (especially an overflow error). Try using smaller time-steps. In a SAPWood-NP analysis, an error message of "Matrix is not stable" will occur if the model is not sufficiently constrained. Please check the model file in this case.

3.4 Unexpected termination during plotting

Generally, this might be the result of trying to plot an all 0 time history. Check if your model has walls in only one direction. If so, the restoring force in the other direction will always be 0 and result in an error in plotting. The rest of the results should still be available.

3.5 Unexpected error during file saving

Check if you have the authority permissions on the computer to create a file in the output path. You can try changing the output path and save again.

4 FAQs

Q: Where can I get the parameters for the walls in my building?
A: If you have wall hysteresis data from a cyclic wall test, it can be loaded into the "manual fit" tool to find the parameters. This will require a basic understanding of the models and parameters for this Version of SAPWood. Another option is to use SAP-NP or the program CASHEW (part of CUREE-Caltech Woodframe research project [Folz and Filiatrault, 2002]) to obtain wall hysteretic parameters from nail parameters and wall configurations. Note that CASHEW assumes the ten-parameter SAWS hysteretic model. In addition, if you do have test hysteresis data which is hard to fit with the ten-parameter model. Please contact the authors or try the EPHM model. The final option is to use the wall parameter database. Keep in mind that the values in the database are for preliminary analysis only.

Q: There's an overflow error, then the program terminates, why will that happen?
A: This might be caused by an unstable model, lack of convergence (especially an overflow error). Try checking the model for instability, using smaller time step.

Q: What does the damping ratio mean in SAPWood?
A: The damping ratio in SAPWood will be used to multiply the critical damping matrix to get a constant damping matrix which will be used during the time domain analysis. The critical damping matrix is a diagonal matrix with its elements calculated as two times the square root of the product of mass matrix and initial stiffness matrix.

Q: What kind of units should I use?
A: The user can choose SI or US Customary/English units in the Environment tab of the SAPWood main frame. This choice defines the value for the gravitational constant G. It is recommended to use either SI units (m, N , Sec) or U.S. Customary units (inch, lb, Sec) when you build your model. The (kips, inch, Sec) is acceptable since the gravity constant (G= 386.4) can still apply. But all the output will be in Kips instead of lb. The same rule applies to kN in SI units. The units note in the output file should be ignored if kips or kN is used.

The earthquake record file should always be in units of gravity, such as 0.2 for 20% of gravity.

Q: Where can I get earthquake input files for my analysis?
A: It is easy to obtain acceleration time history data for historical earthquakes from the USGS website and other strong motion websites. However, the user may need to generate the *.eq2 file using other programs (MATLAB or Excel, for example). The format of the *.eq2 file is as shown below (if the earthquake has n data points):
t1 ax1 ay1 az1
t2 ax2 ay2 az2
t3 ax3 ay3 az3
…
…

tn axn ayn azn
There are 20 earthquake record files in the SAPWood installation package which are ready to use once the user installs SAPWood. An earthquake folder will be created in the installation folder to contain these files.

Q: Where can I get the displacement protocol files?
A: It is easy to generate the *.pro file using other programs (MATLAB, Excel, or even a text editor for example). The format of a *.pro file is as shown below (if the protocol has n data points):
x1
x2
x3
…
…
xn

Note that the protocol file can be very "coarse" (containing only the key points). The user can use a large sub-step number in the analysis to get a refined result.

Q: Everything is in PGA here, is there a way to use Sa (spectral acceleration)?
A: There's an option to use Sa in the environment tab (the user will also need to specify the target/spectral period and damping ratio for the Sa). If this is option is selected, all the places in this users manual where PGA appears will simply appear as Sa instead. The program will automatically scale the earthquake to the specified Sa before performing the time domain analysis. For the situation of bi-axial excitation where the excitations in X and Y differ, the scaling will be done separately for both directions.

However, for hazard environmental file used in loss simulation, the user needs to specify in this file whether PGA or SA is used. The analysis will not be affected by the PGA-Sa option in the SAPWood main window tab.

Q: What is the unit system used in SAPWood?
A: The program itself is without any units. But when the user selects SI or US from the environment tab, the gravity constant G will be set to 9.81 or 386.4 based on the selection. So please be sure to use consistent units in the model files to avoid errors that are difficult to detect.

Q: What does the global displacement and rotation really mean in SAPWood?
A: The global displacement is measured relative to the non-shaking ground at the centroid of the first floor diaphragm. For upper stories, the projectile location of the first floor centroid is used. The rotation is relative to this point, NOT the origin of the coordinate system which the user used to create the floor plan input. This will make sure that you get the same answer no matter how you created your model. However, you can manually override this option by assigning 1 to the coordinate option data in the model file. In that case, the displacement and rotation will be measured from the origin of the coordinate system in which the nodes of floor diaphragms are defined.

Q: Where were all my output files and model files saved to?
A: It is suggested that the first step upon opening SAPWood should be to change the output directory setting to the directory you would like your files to be directed. Otherwise, ALL the output files (including

the model file constructed by model builder) will be saved to C:\ by default.

Q: I load the model into the workspace, but can't find it in the analysis window's structure list - what should I do?

A: The analysis window's structure list only refreshes upon loading. This means that the model loaded after the analysis window is loaded will not appear in the list. The simplest way to solve this is to close the analysis window and reload it from the main menu. Note that you do not need to load any model in the workspace to do Multi-case IDA. MIDA directly load the model from the file locations you specified.

Q: I loaded a SAWS file and try to get a plot of wall locations, it doesn't look right. What happened?

A: The SAWS input file does not have the wall length and complete location (it only has 1 coordinate) information. So, the SAPWood program assigns default values for this missing information. The "plot" option in the story property window will be plotted based on this default information, thus its appearance will not be what is expected. However, you can load the SAWS file, update the information of each wall using the "edit mode", and save the modified model to a SAPWood model.

Q: I made some changes for the model in the workspace, how can I save it?

A: Go to File->New model and bring up the model builder, then use the "Import" option to load the model in the workspace into model builder. Then simply save it to any file name you would like.

Q: Why can't I save my "half-completed" model? Why do I need to start all over when construct a model?

A: This cannot be achieved if you use the interactive model builders. The best solution is to follow the model file format instruction and establish the model in a text editor (notice one can use Excel sheet to help writing and cut-paste the page into a text editor, which make it easier to follow the format).

Q: What unit system should I use for my SAPWood-NP model?

A: You must use the unit system which is consistent with the unit system used in your nail parameter data.

Q: What coordinate system should I use for my SAPWood-NP model?

A: Use a global coordinate system of your choice for model construction. However, the system must have its X axis in the horizontal direction and Y axis in vertical direction of the shearwall you build.

Q: SAPWood-NP model builder always tells me I have repeated nails, what happened?

A: An NP model will not allow 2 nails at the exactly same location. If you do want to put the nails in such a configuration, shift one of the nails a little bit, such as 0.01 inch. Note that this may be occurring at the corner nails.

Q: The SAPWood-NP analysis yields unusual/unrealistic wall hysteresis loops, and the deformed wall plot doesn't make sense, what happened?

A: If the user wants to impose a displacement beyond the wall capacity, the model will become unstable. If the wall model is not constrained properly (such as one component is not connected to any of the others by mistake), the model will become unstable. However, the program can still provide the results based on numerical evaluation which may make no practical sense. Some suggestions to avoid this situation are: Use a monotonic protocol first to find out the limit of the wall; avoid cyclic protocols with too many cycles

since the numerical error may accumulate; constrain the wall components properly; use the appropriate number of sub steps.

Q: How do I decide the length of a wall segment to be built as "one wall element" in SAPWood NP?
A: The NP model can consist of a very long segment of shearwall (even with holes and openings) or a short solid wall segment. The choice will affect the final outcome of the assembly level (house) model (fewer wall elements representing very long wall segments or more wall elements obtained from shorter wall segments). With the final product (assembly level model) in mind, the user can choose the configuration accordingly. However, it is recommended that one NP model have ONE top stud so that the model actually acts as a SINGLE wall element. Notice that for some of the long wall segments with openings; it is not possible to build the wall with one top stud. But as long as the top of the wall deforms as a single stud during cyclic loading, it can be modeled with one top stud in the NP model. So, it is good practice to use short wall segments if one is not sure about the wall behavior. In addition, these shorter segments have a higher likelihood of being used in future models since they are typically more general.

Q: I do not understand the theory behind the loss estimation procedures. Is there anything I can use as a reference?
A: Please see the reference section of this manual.

Q: I am not familiar with loss estimation and have never done it before. How hard would it be for me to use the loss analysis tool in SAPWood to perform analysis on my structure model?
A: Unfortunately, the user of the loss analysis in SAPWood will need a considerable amount of background and preparation to effectively use the software. Some knowledge of this topic is needed to construct the uncertainty models needed for the simulation. A thorough review of the loss simulation framework behind the program is likely needed before one can make full use of it. However, as the loss estimation procedure gets more mature in future research, the ultimate goal for the loss analysis module in SAPWood is to provide a loss estimation tool that can be used by an analyst with limited expertise in uncertainty modeling and loss estimation. A future release of SAPWood will provide this capability.

5 Examples

This example will illustrate the use of some of the SAPWood tools by going through three analyses. The first one is the analysis of a two story woodframe townhouse. The second one is the cyclic analysis of a simple wood shearwall. The third one is the long term loss simulation of a residential building. The model files (structure model and wall model), earthquake and protocol files can be found in the distribution package (in the installation directory after installing SAPWood). One of the models used here is a simplified model for the NEESWood benchmark structure tested at the University at Buffalo (NEES@buffalo) NEES laboratory in 2006 as part of the NEESWood Project (Christovasilis et al., 2007). The floor plans of the benchmark structure are attached in the Appendix. The residential structure is a typical single family home whose floor plan is also attached in the Appendix.

5.1 Benchmark structure analysis

5.1.1 Building the model

Step 1: Load SAPWood, open the model builder.

Step 2: Input the number of stories, and number of wall elements in each story, click on "Set" to generate the empty model structure.

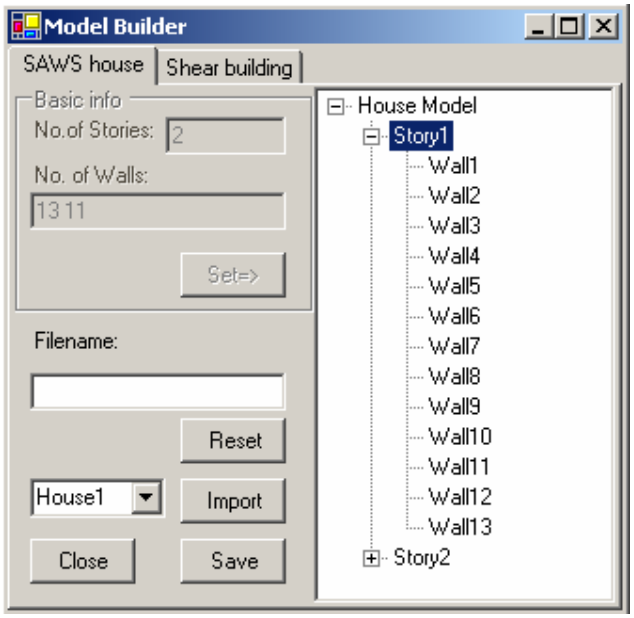


Figure 21. Building a two story townhouse model

Step 3: Double-click on wall 1 of story 1, bring up the wall property window.

Step 4: Choose wall direction, input coordinates for wall center (X,Y), enter the wall mass and rotational mass.

Step 5: Choose the model for the wall, the parameter tips will appear above the parameter input box, then input the parameters following the sequence indicated by the parameter tips, separate each parameter by a blank.

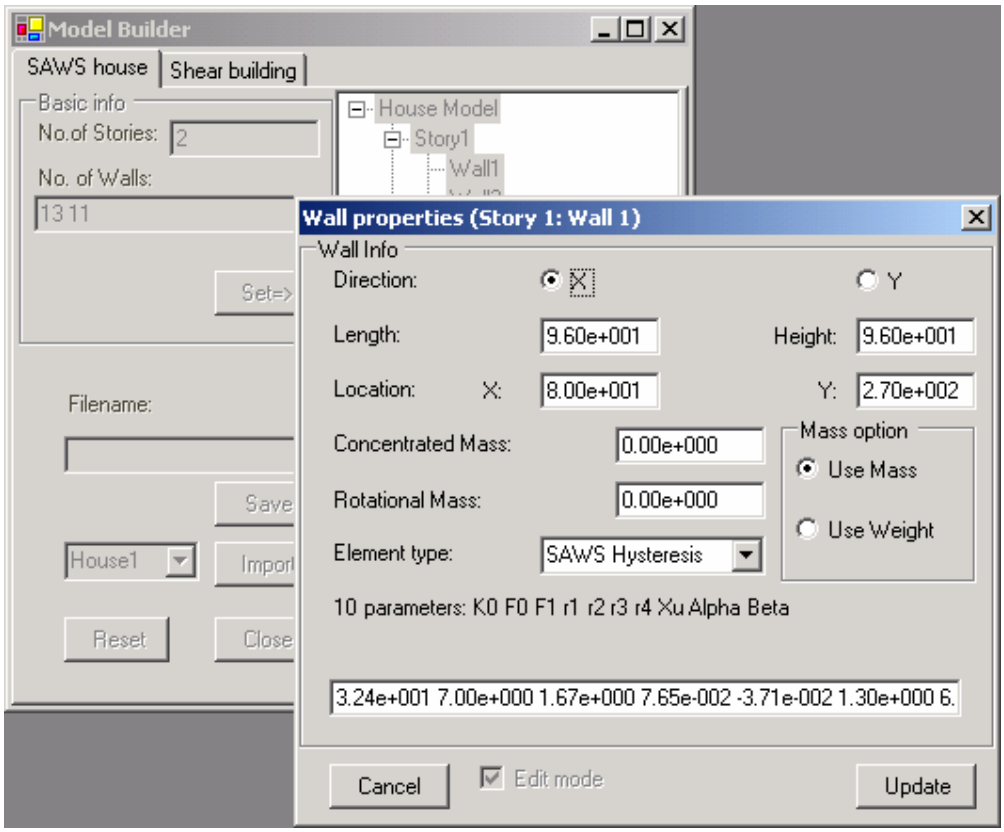


Figure 22. Input wall parameters

Step 6: Click "Update" to close the window and go back to model builder.

Step 7: Repeat step 3-6 for all other walls

Step 8: Double click on story1, bring up the story property window.

Step 9: Input the Nodal coordinates in a counter clockwise sequence, "Add" each node to the list.

Step 10: Input the distributed mass value. Note that it is with units (mass/area).

Step 11: Input the concentrated masses on this floor, including the coordinates (X,Y), the mass (M), and the rotational mass (RM). Add them into the list.

Step 12: Click on "Plot and Calculate" to check your input from the plot (the yellow square is the location of Centroid, the red circle is the location of concentrated mass). Also check the total mass (Area x distributed mass).

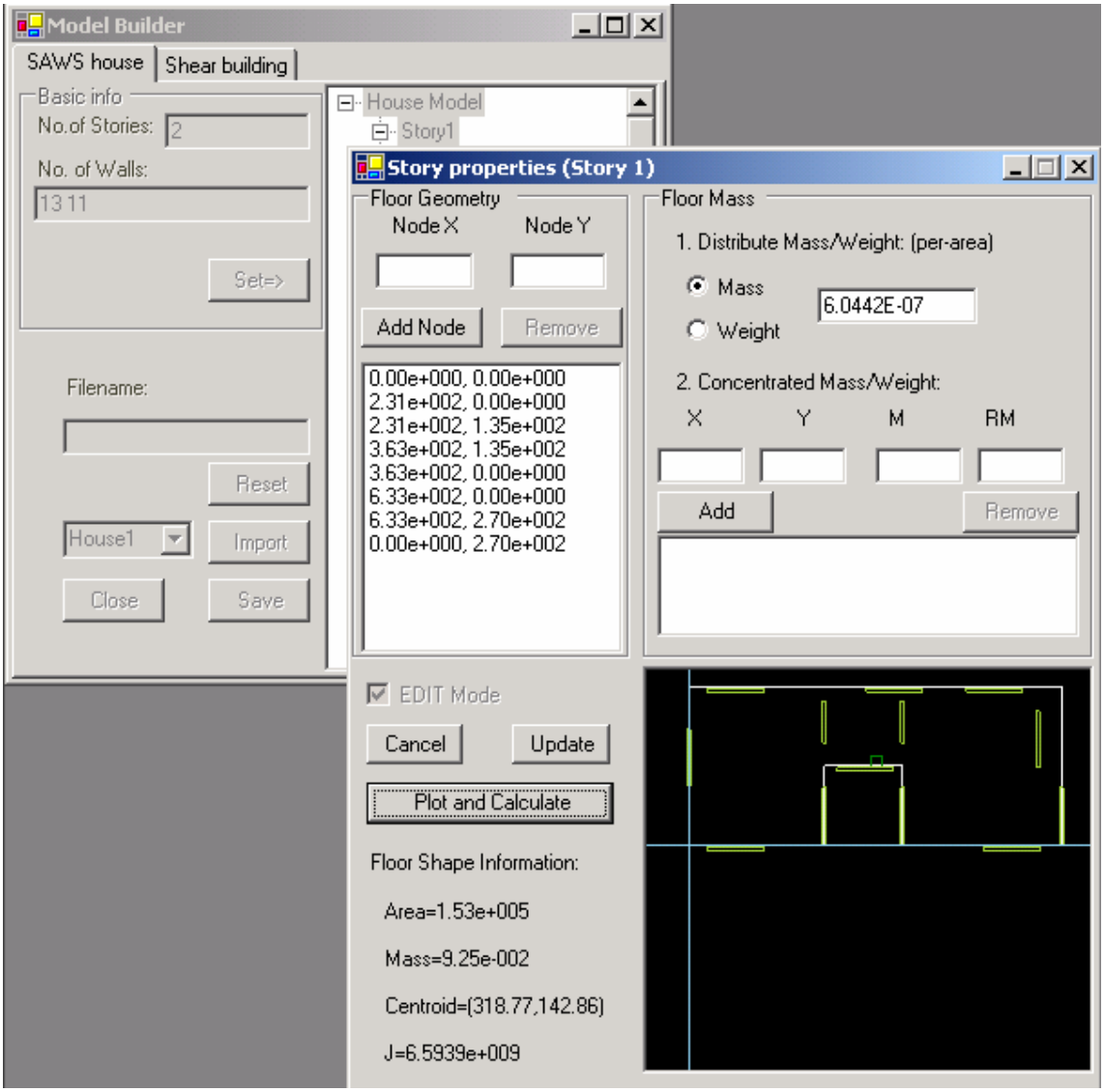


Figure 23. Input and check the floor configuration

Step 13: If everything is correct, click "Update" to close the window and go back to the model builder.

Step 14: Repeat step 8-13 for story 2

Step 15: Input a file name you want this model file to be saved as. (Here we use "BenchMark")

Step 16: Go to the Environment tab of the Main frame, change the output directory to the directory where you want this file to be saved to. (Here we use "u:\temp\")

Step 17: Click "Save" and a message box will tell you the file has been saved as "filename.sap"

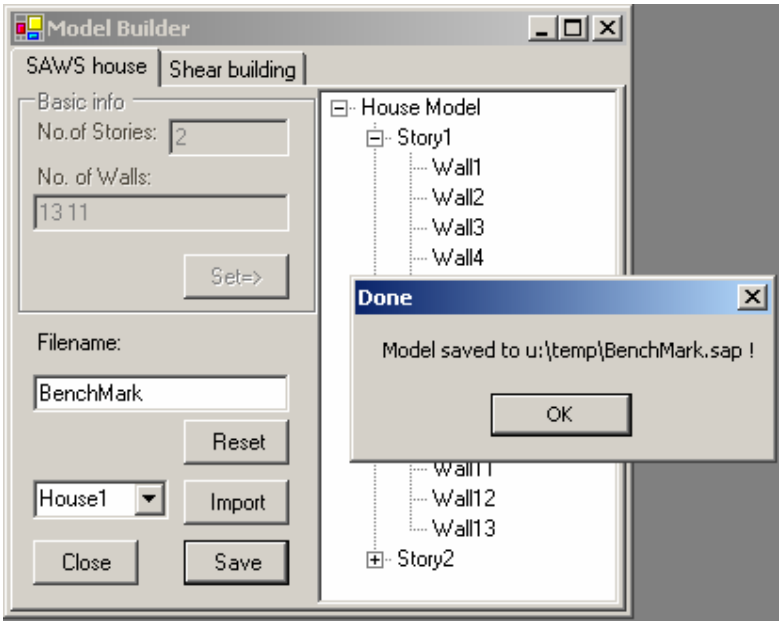


Figure 24. Save to a model file

5.1.2 Single earthquake excitation analysis

Step 1: Click "File-> load model" to find and load the file you just created in section 1.

Step 2: (Optional): In the work space, select the wall or story and use "Information" button to bring up property window to check the walls and stories in your model to see if it is the model you want. (Note you can also edit the properties of the model in the work space by enabling "EDIT mode".)

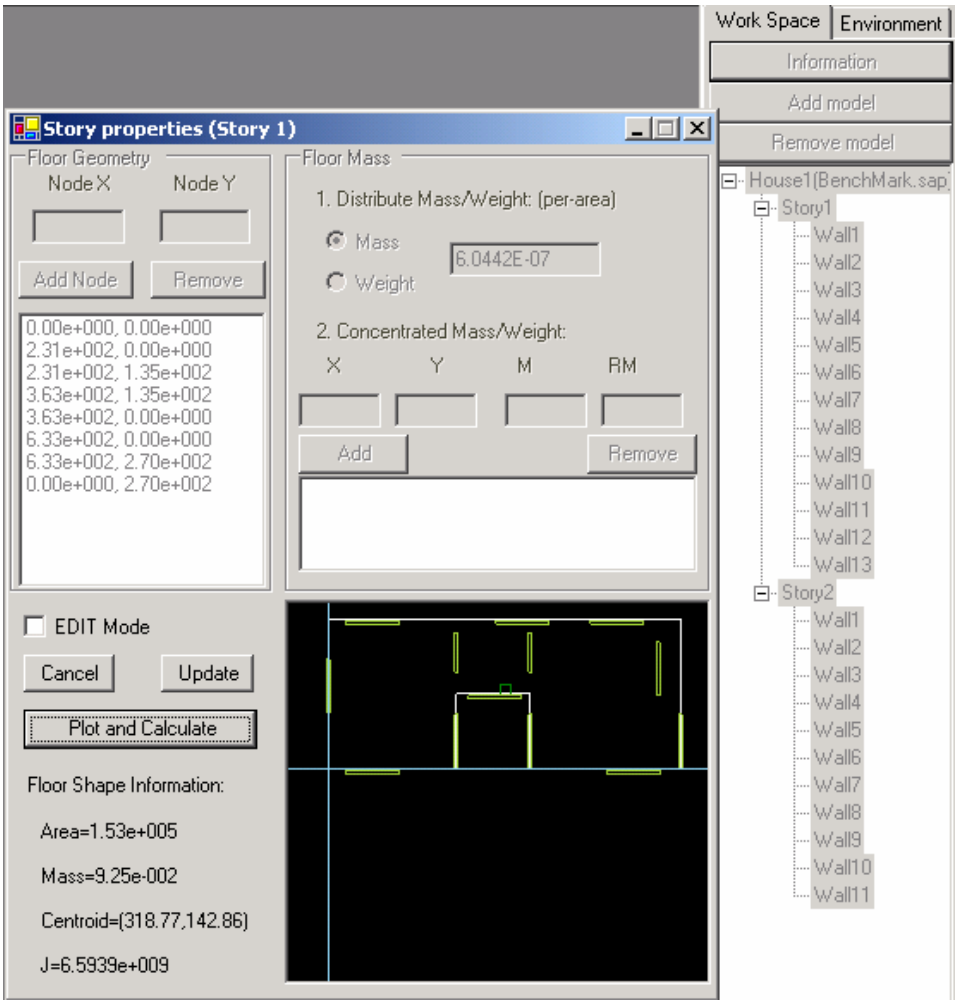


Figure 25. Load the model, view it in workspace

Step 3: Load the Single Excitation analysis tool by clicking "Analysis->EQ excitation".

Step 4: Load the earthquake record file (*.eq2) by clicking the "Load" button. (Use Acm1.eq2 for the example)

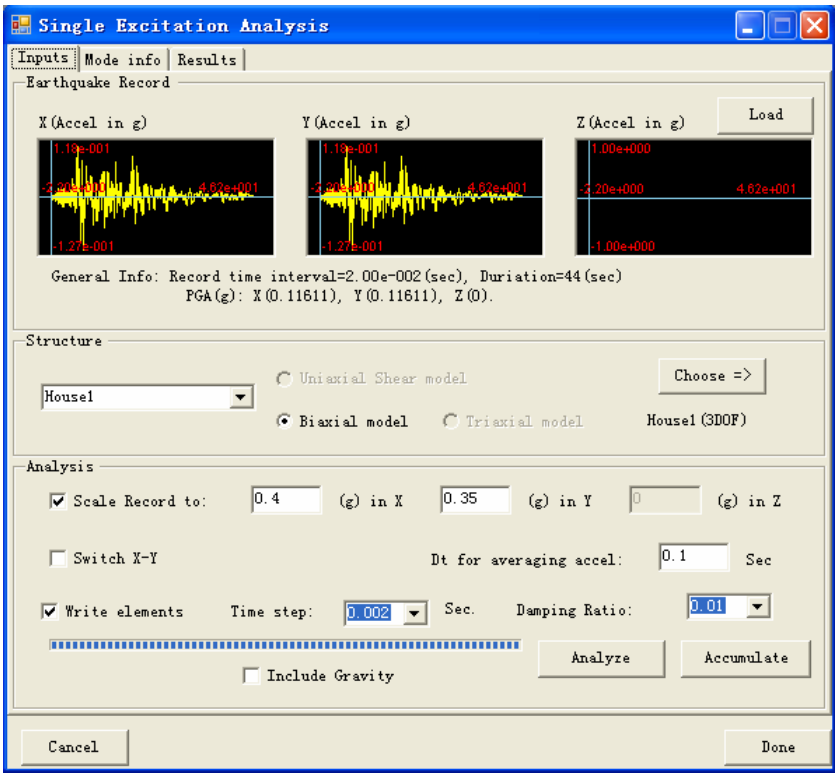
Step 5: Choose the model from the pull-down list and click "Choose"

Step 6: Click the "Scale Earthquake to" option and put in the value of PGA in X and Y in unit of gravity (Here we use 0.5 and 0.4).

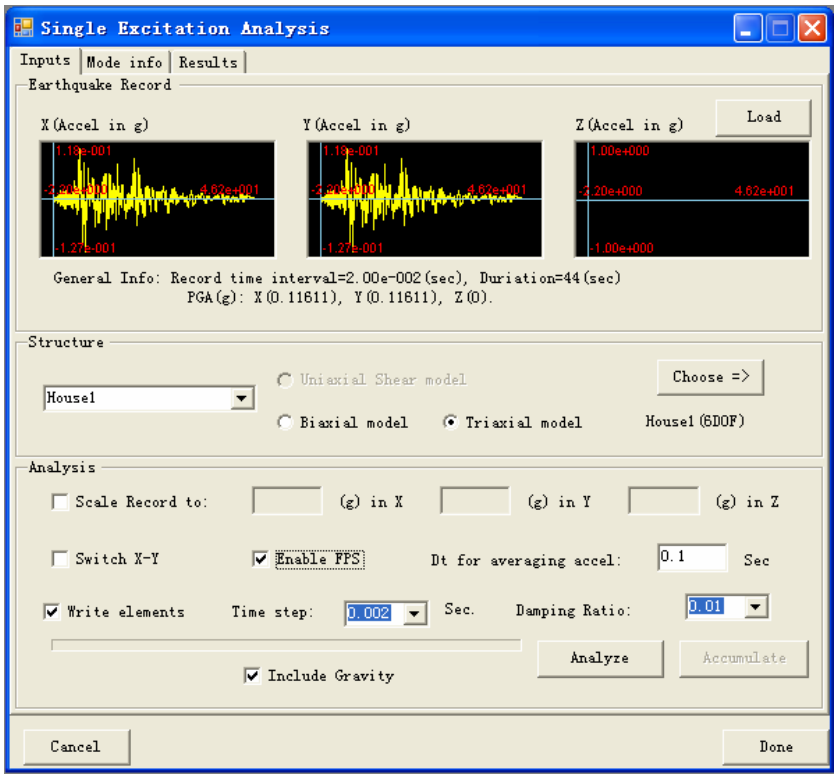
Step 7: Take a look at the "EQ Dt" at "General Info" of earthquake you just loaded. Assign time step value of 0.002 sec.

Step 8: Select or enter a damping ratio of your choice. (Here we use 0.01)

Step 9: If a triaxial model is loaded with FPS option, the "enable FPS" box will appear (Figure 26 (b)). The user can enable or disable the FPS to see the difference in responses.



(a)



(b)

Figure 26. Run bi-axial excitation

Step 10: Click on "Analyze". After the analysis is done, click on "Accumulate". This is to successively load the structure with the same earthquake twice.

Step 11: Wait for the end of analysis. Go to the "Mode info" tab, click on "Calculate", the natural period, fundamental mode, and dynamic matrix appears.

Step 12: Go to the "Results" tab, select the Global DOF from the pull-down list, then the time history is automatically plotted.

Step 13: Select any wall from the pull-down list, then the wall hysteresis is automatically plotted. (The wall number here is consistent with the wall number in your model file.)

Step 14: Repeat step 11-12 for all the responses of interest.

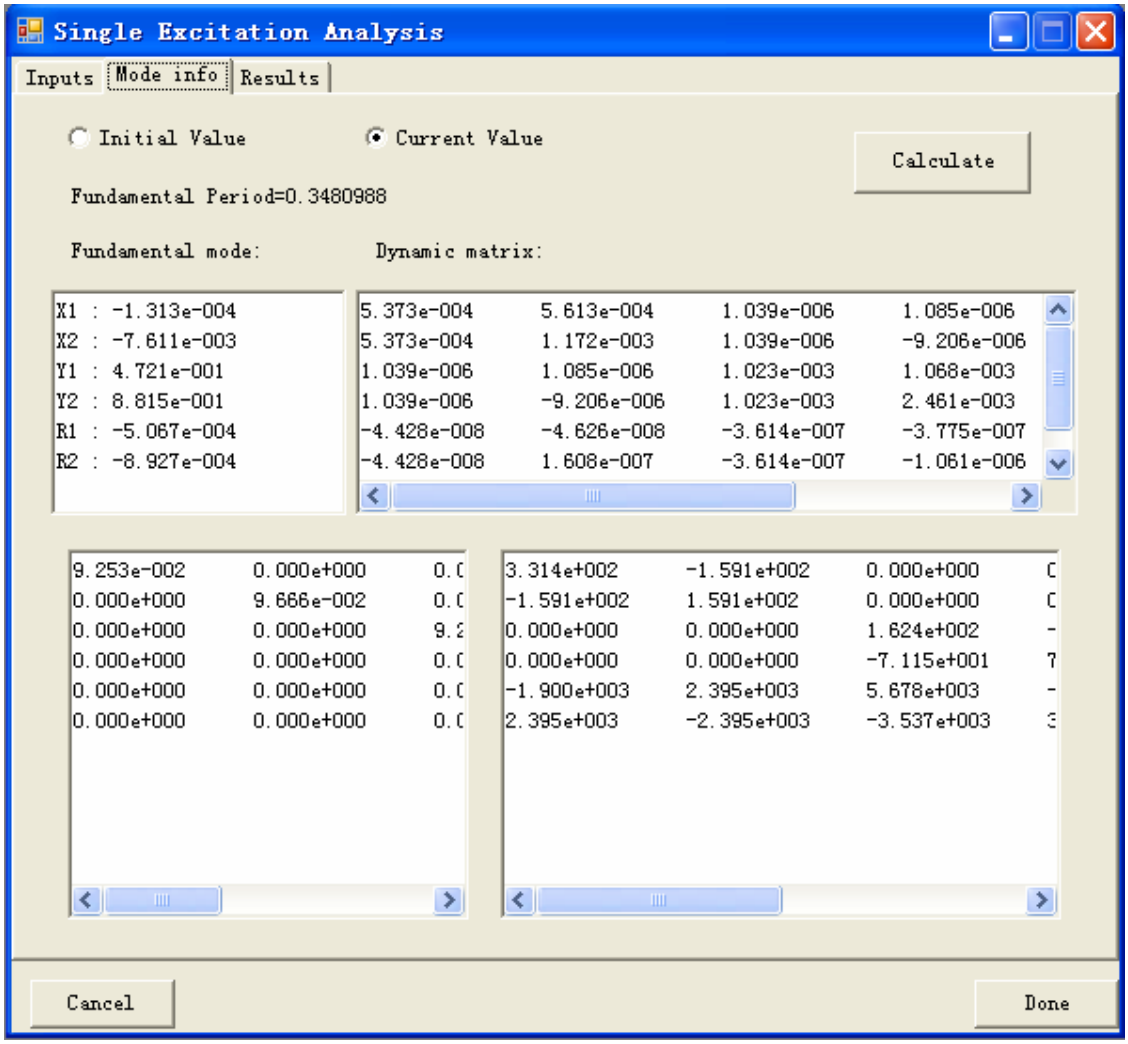
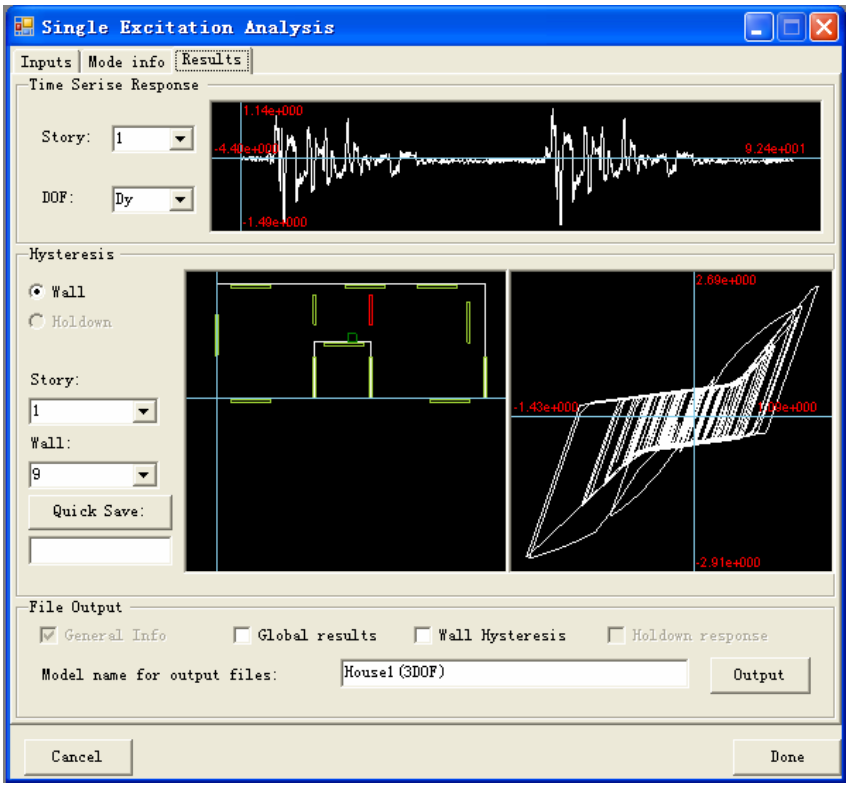


Figure 27. View the modal results from analysis

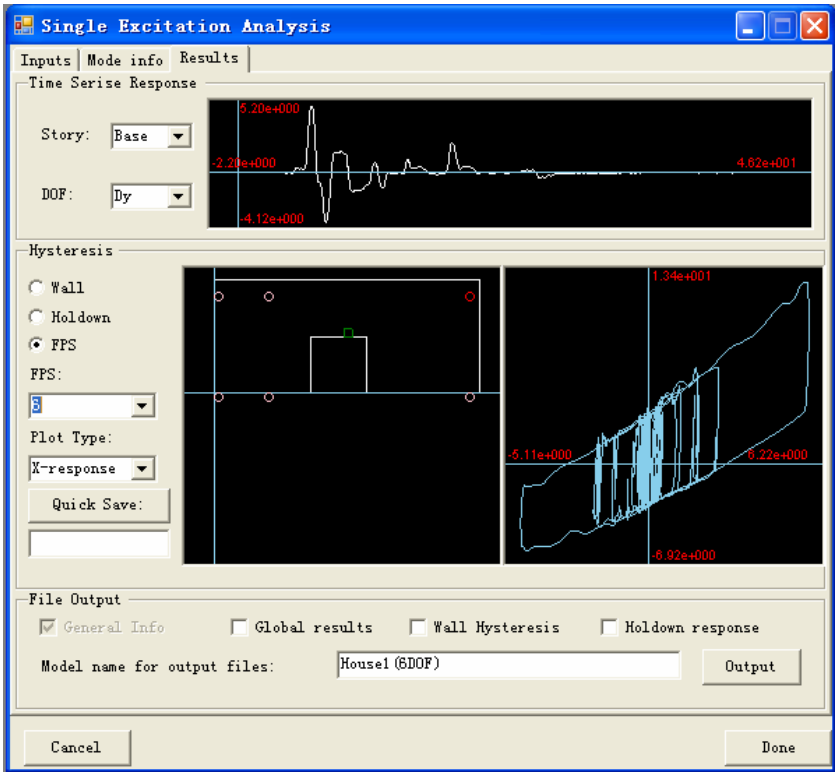
Step 15: Click the file output options you need, then enter the filename you wish to save them under. (Here we use "MyHouse").

Step 16: Go to the Environment tab of the Main frame, change the output directory to the directory where you want these files to be saved to.

Step 17: Click "Output" and the results will be saved in the files of your choice. (Here the files are: MyHouse.gen, MyHouse.Glo, and MyHouse.hys)



(a)



(b)

Figure 28. The viewing and output of the results

If a triaxial model (BenchmarkTriaxial.sap) with FPS option is used in the analysis, the response of the

FPS isolator can also be observed after the analysis, as it is shown in Figure 28 (b).

5.1.3 IDA analysis

Step 1: (Skip to step 2 if model is already in the workspace): Click "File-> load model" to find and load the file you just created in section 1.
Step 2: (Optional): In the work space, use "Properties" to check the walls and stories in your model to see if it is the model you want.
Step 3: Load the Incremental Dynamic analysis tool by clicking "Analysis->IDA".
Step 4: Load the earthquake record file (*.eq2) by clicking the "Load" button.
Step 5: Choose the model from the pull-down list and click "Choose"
Step 6: Select a bi-axial option. If "Customized X:Y" is chosen, input PGA-x/PGA-y or Sa-x/Sa-y value in the blank followed. (Here we use a ratio of 1.5).
Step 7: Add the earthquake PGA's you want to analyze to the PGA or Sa vector by enter the values one by one and click "Add".
Step 8: Set time step and damping ratio in the same way as in Single earthquake excitation analysis.

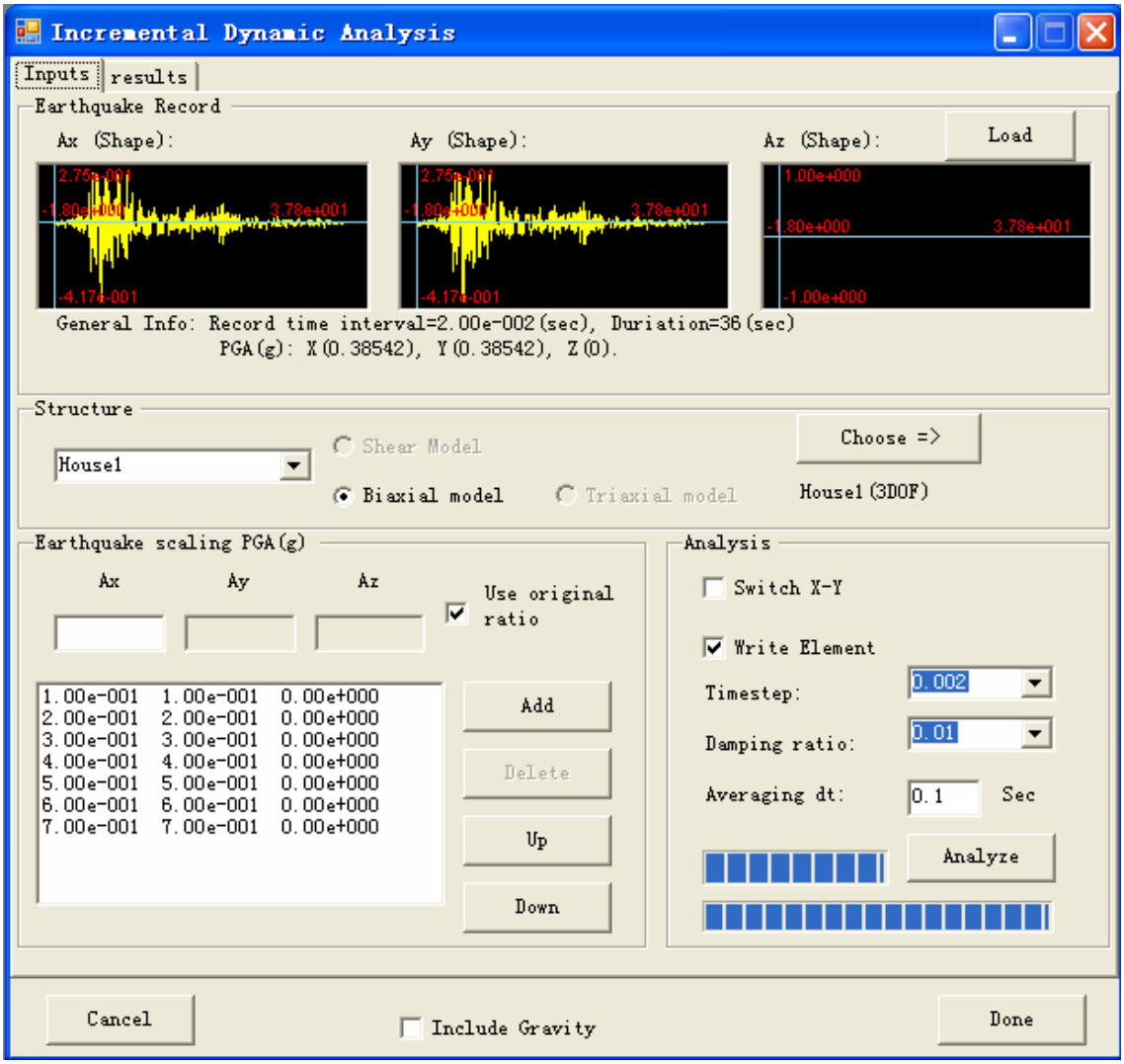


Figure 29. Setup for the IDA analysis

Step 9: Click "Analyze" and let the program run.

Step 10: Wait for the end of analysis, then go to the "Results" tab.

Step 11: Select the Global DOF from the pull-down list, and the IDA curve is automatically plotted. The points on the curve are also listed at the same time.

Step 12: Select any wall from the pull-down list, then the wall IDA curve is automatically plotted. The points on the curve are also listed at the same time.

Step 13: Repeat step 11-12 for all the responses of interest.

Step 14: Enter the Model name you wish to save them under. (Here we use "MyHouseIDA").

Step 15: Go to the Environment tab of the Main frame, change the output directory to the directory where you want these files to be saved to.

Step 16: Click "Output" and all the results will be saved in the files of your choice. (Here the file is MyHouseIDA.IDA).

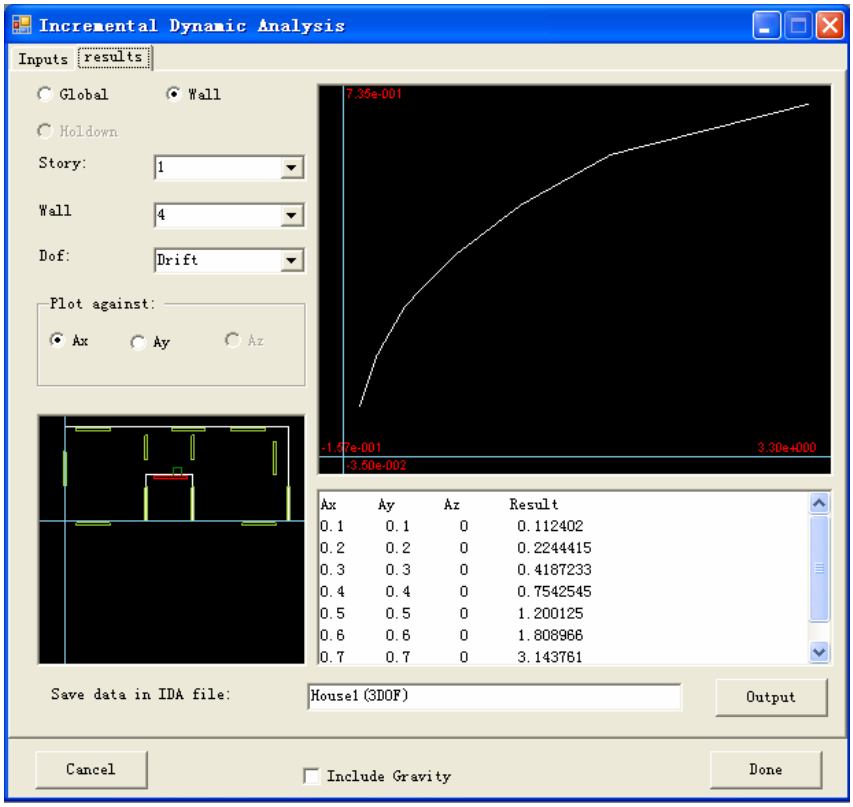


Figure 30. View IDA curves and results

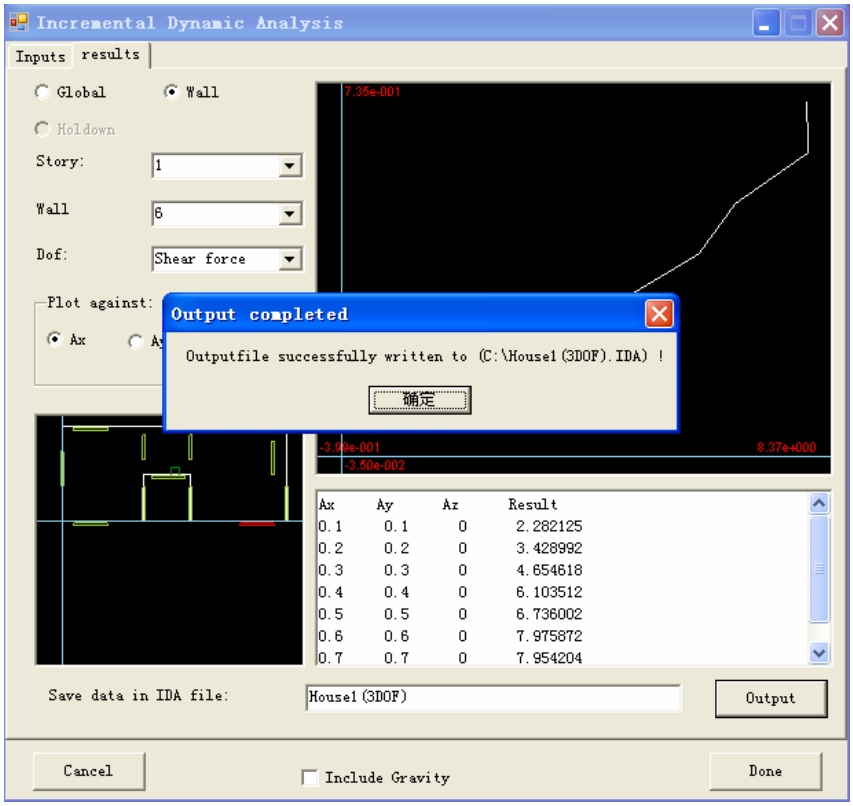


Figure 31. Save IDA's to file for future use

5.1.4 SDOF Identification analysis

Step 1: (Skip to step 2 if the model is already in the workspace): Click "File-> load model" to find and load the file you just created in section 1.

Step 2: (Optional): In the work space, use "Properties" to check the walls and stories in your model to see if it is the model you want.

Step 3: Load the SDoF Identification analysis tool (Cyclic Pushing Analysis) by clicking "Analysis->SDoF Identify".

Step 4: Load the protocol file (*.pro) by clicking the "Load" button. (protocol_random.pro)

Step 5: Choose the model from the pull-down list and click "Choose"

Step 6: Click the "Scale to Max=" option and put in the value the desired maximum value of the protocol (Here we use 2.3).

Step 7: Take a look at the "maximum step" at "Info" for the protocol you just loaded. If that number is big (greater than 0.01 inch), you may want to assign a "Substep" number greater than 1 so that the actual step in analysis is small enough. (Here we use Substep=50). This is a fairly quick analysis, so a large number of sub-steps is recommended for accuracy.

Step 8: Choose the degree of freedom you want this analysis to be applied to. (Here we use the Dx of the 1st story).

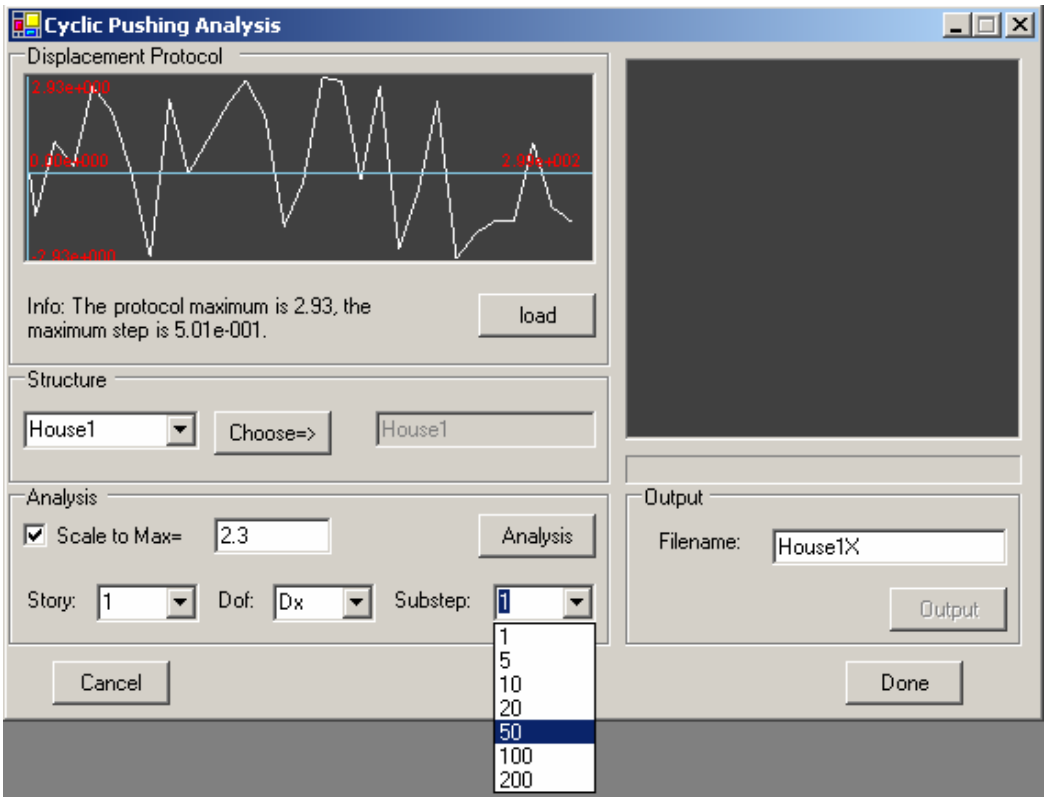


Figure 32. Prepare inputs for SDOF identification

Step 9: Click "Analysis". The results will appear after the analysis is completed.

Step 10: Enter a filename (here we use "House1X") to output the resulting hysteresis to a file.

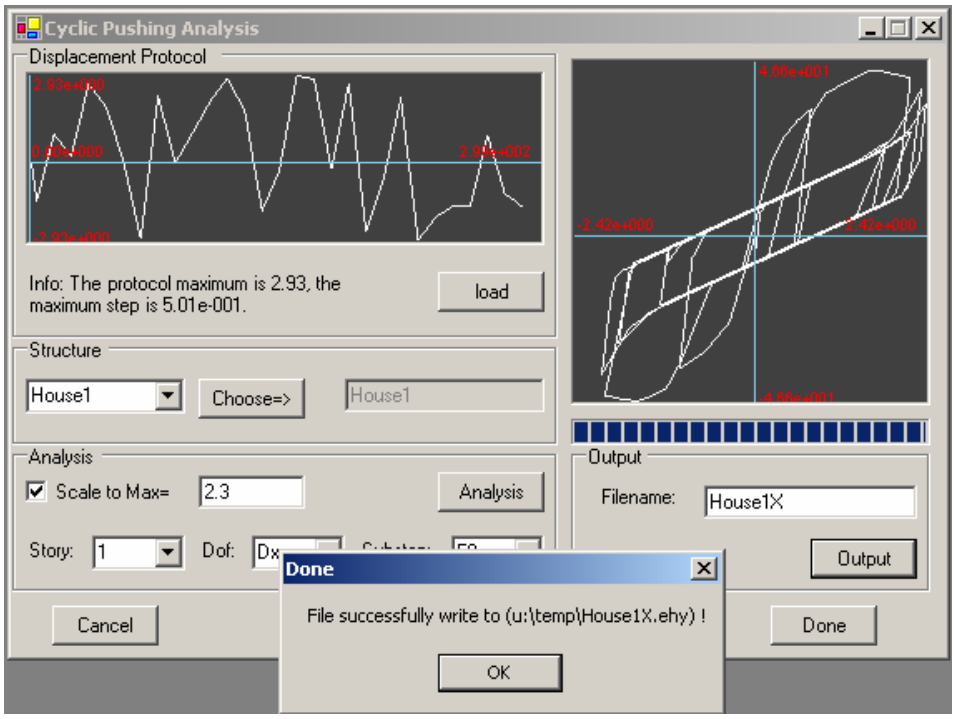


Figure 33. View and output results

5.2 SAPWood-NP wall model analysis

5.2.1 NP model building

Step 1: Open NP model builder.

Step 2: Enter 6 studs and 1 panel for Basic info and press "Set".

Step 3: Enter the coordinates for each stud and panel as shown in Figure 31. Set "Fixed" condition to bottom stud.

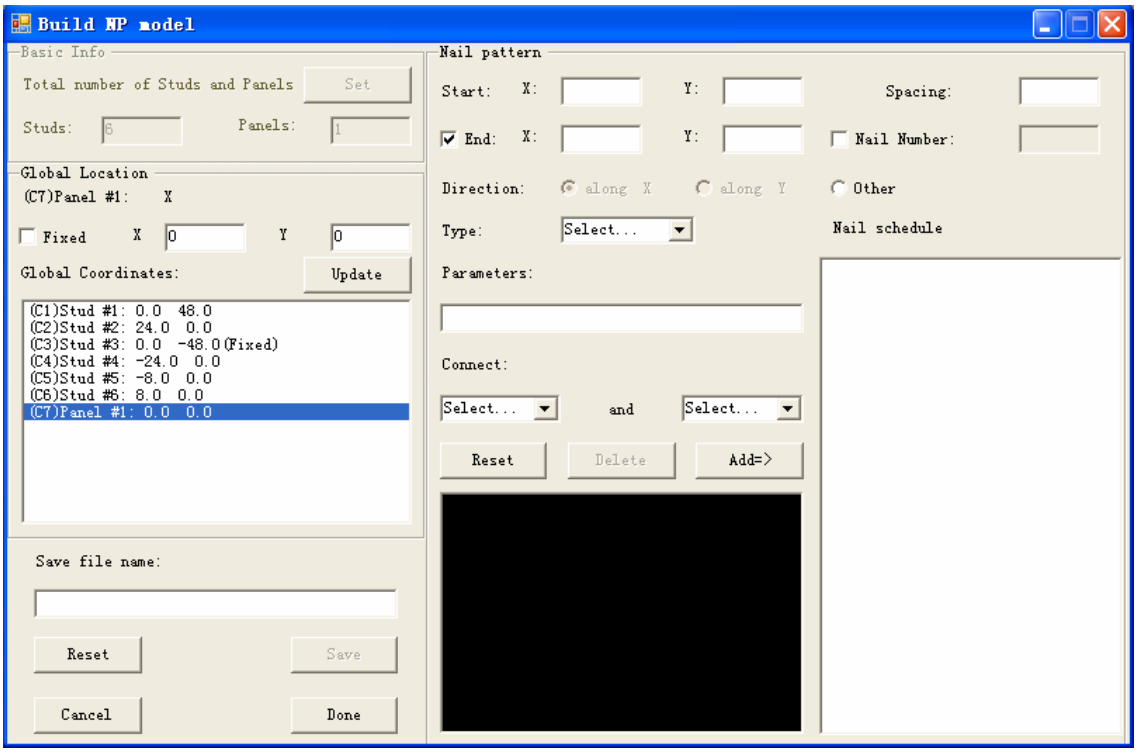


Figure 34. Create NP model (input components info)

Step 4: Input nail line information. First line will be starting at (-24, 48), ending at (24, 48), spaced at 6 inches. The Nail Type will be SAWS hysteresis here with parameters:

2000 500 50 0.0001 -0.1 1 0.015 0.35 0.75 1.1

And the nail line connects stud #1 and panel #1.

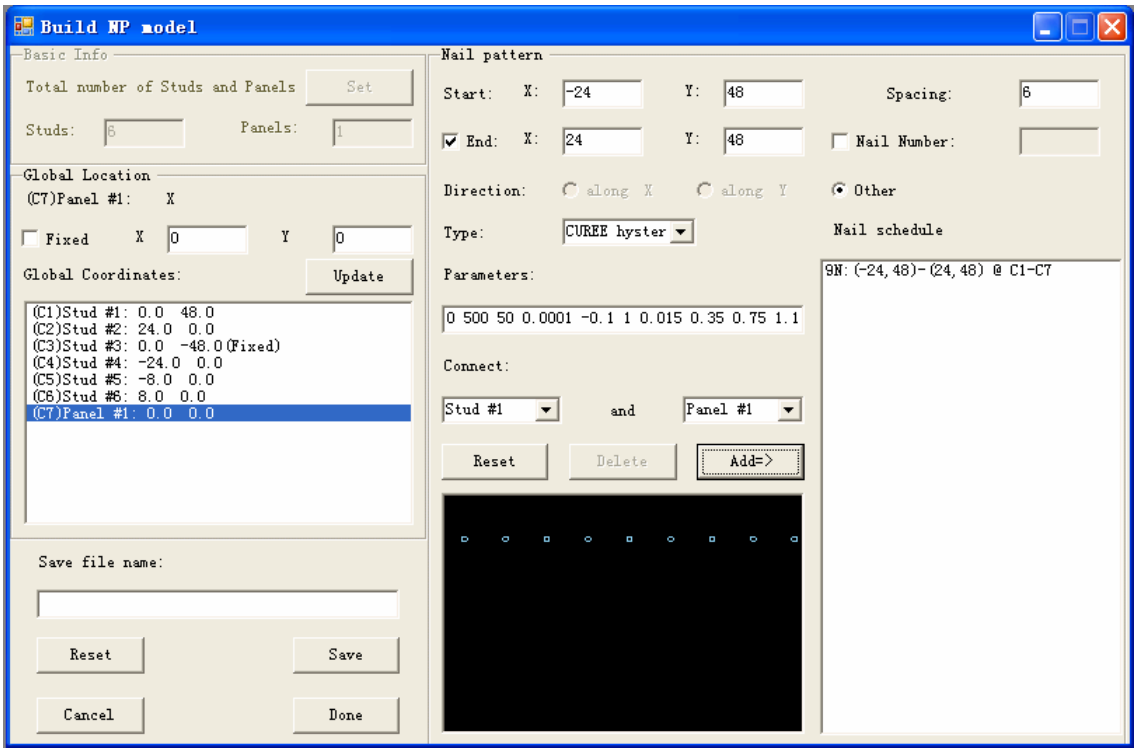


Figure 35. Create NP model (first nail line info)

Step 5: Continue entering all other nail lines.

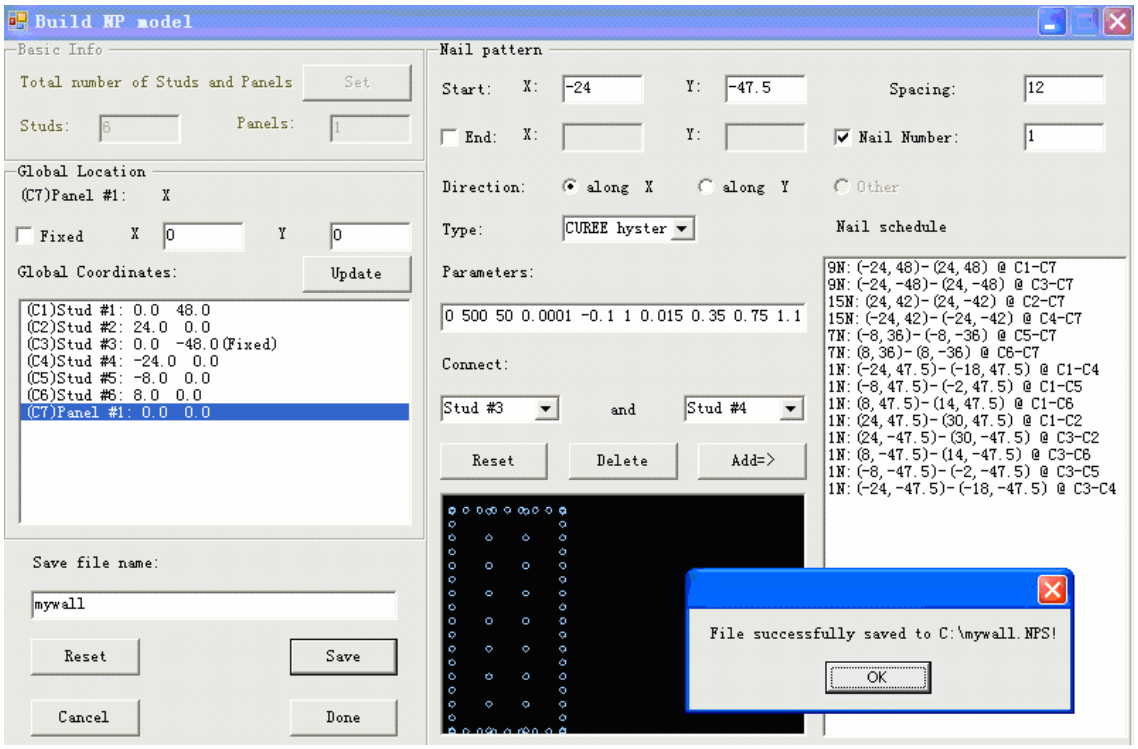


Figure 36. Create NP model (completed and saved)

Step 6: Save the model as "mywall.NPS".

5.2.2 NP model analysis

Step 1: Open NP model analysis window.
Step 2: Load cyclic loading protocol "protocol_cyclic.pro".
Step 3: Load the model just created "mywall.NPS".
Step 4: Scale protocol to a maximum displacement of 4 inches. Use sub step=1, set Pushing DoF to "1X", check "Check nail" box and run analysis.

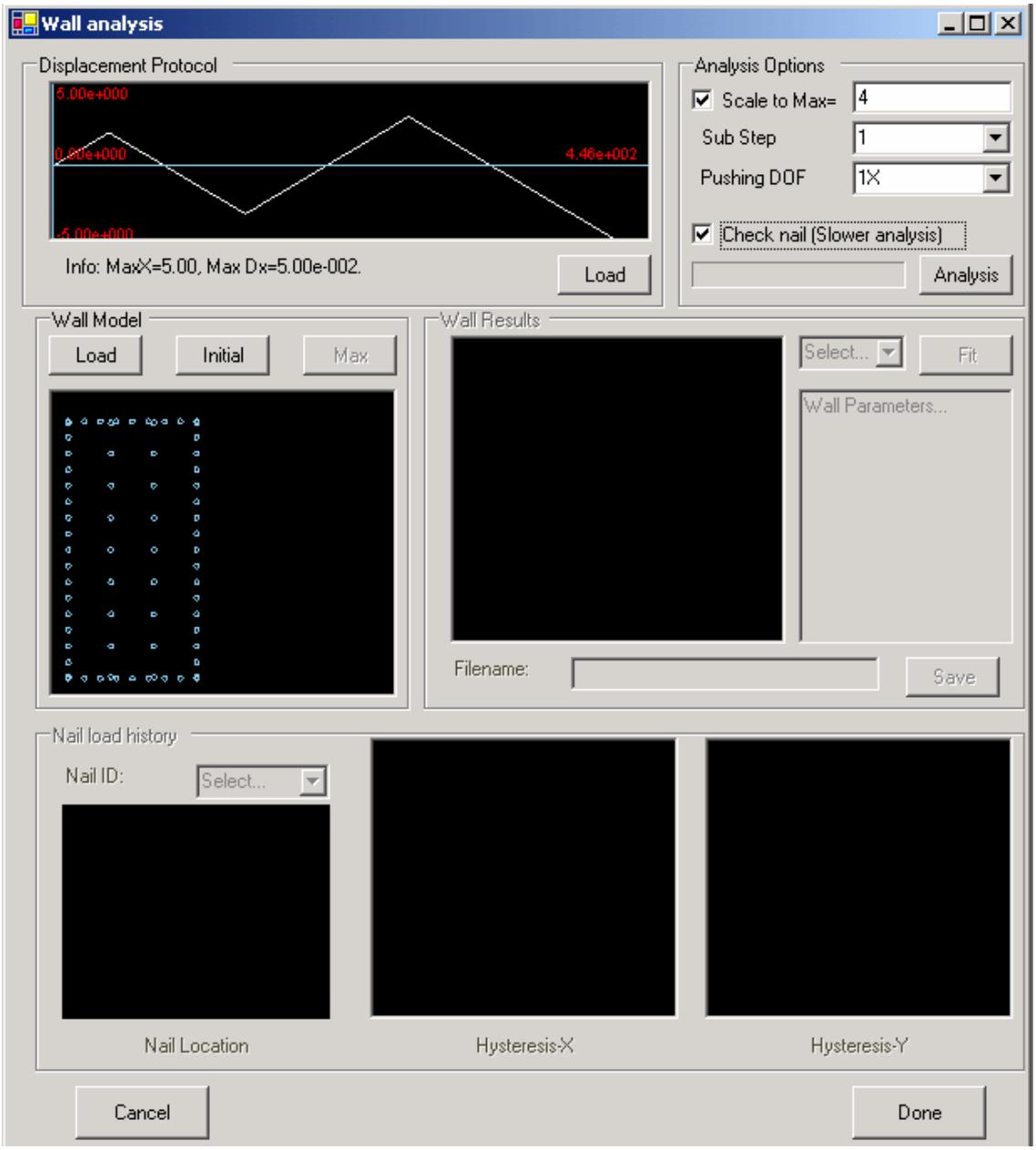


Figure 37. Load NP model and protocol

Step 5: Results will appear after analysis is completed. And the nail information can also be viewed by choosing the nail ID (the nail of choice will be plotted in red also.).

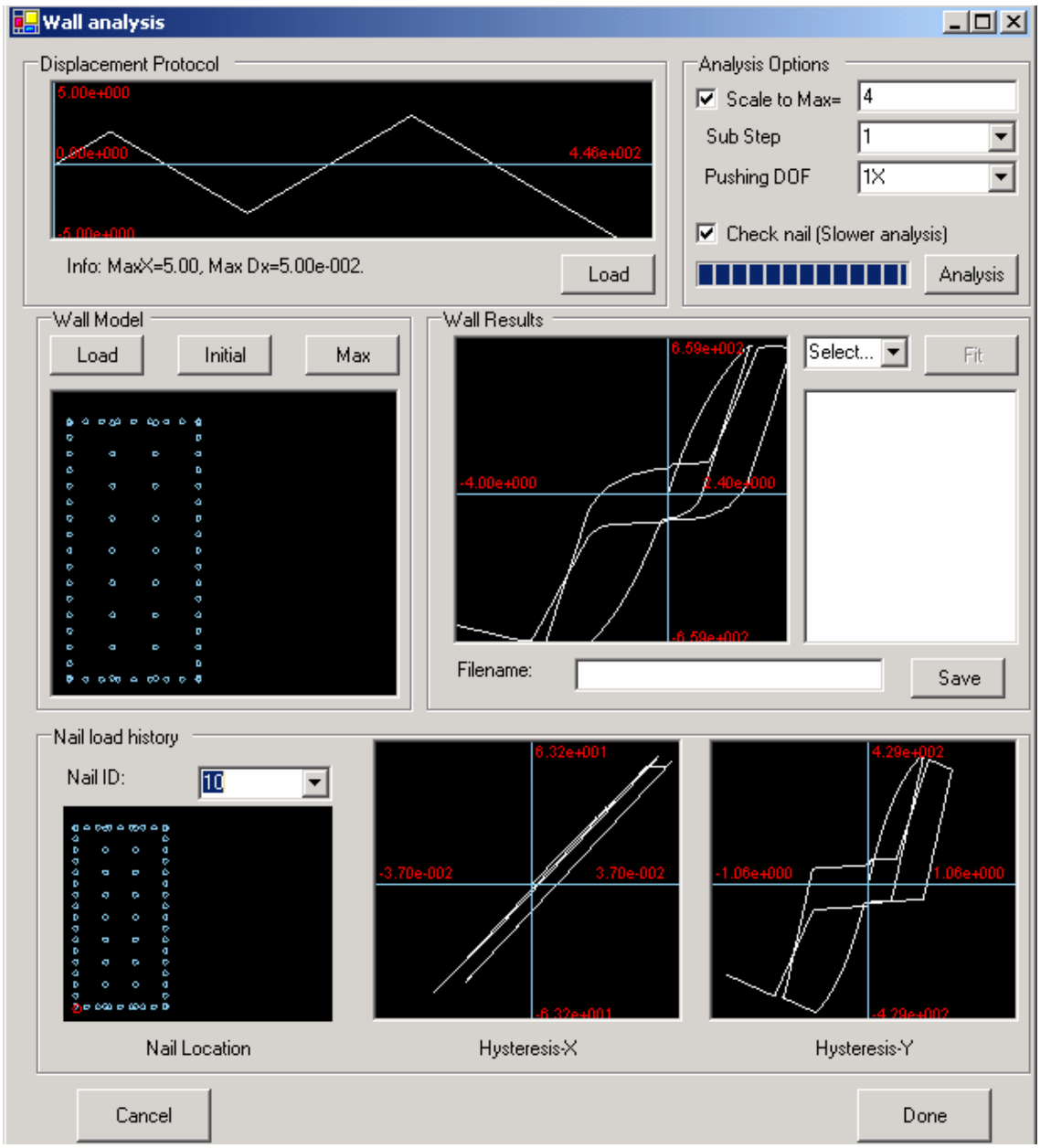


Figure 38. Perform NP cyclic loading analysis

Step 6: Choose "CUREE hysteresis" to fit the wall behavior, press "Fit" and the fitted model will appear on the same plot. The fit is not as good as we would like.

Step 7: Change K0 to 1000 and r2 to -0.05 and hit "Fit" again. A better fitting result appears. This set of parameters can be used to represent this wall in a house model.

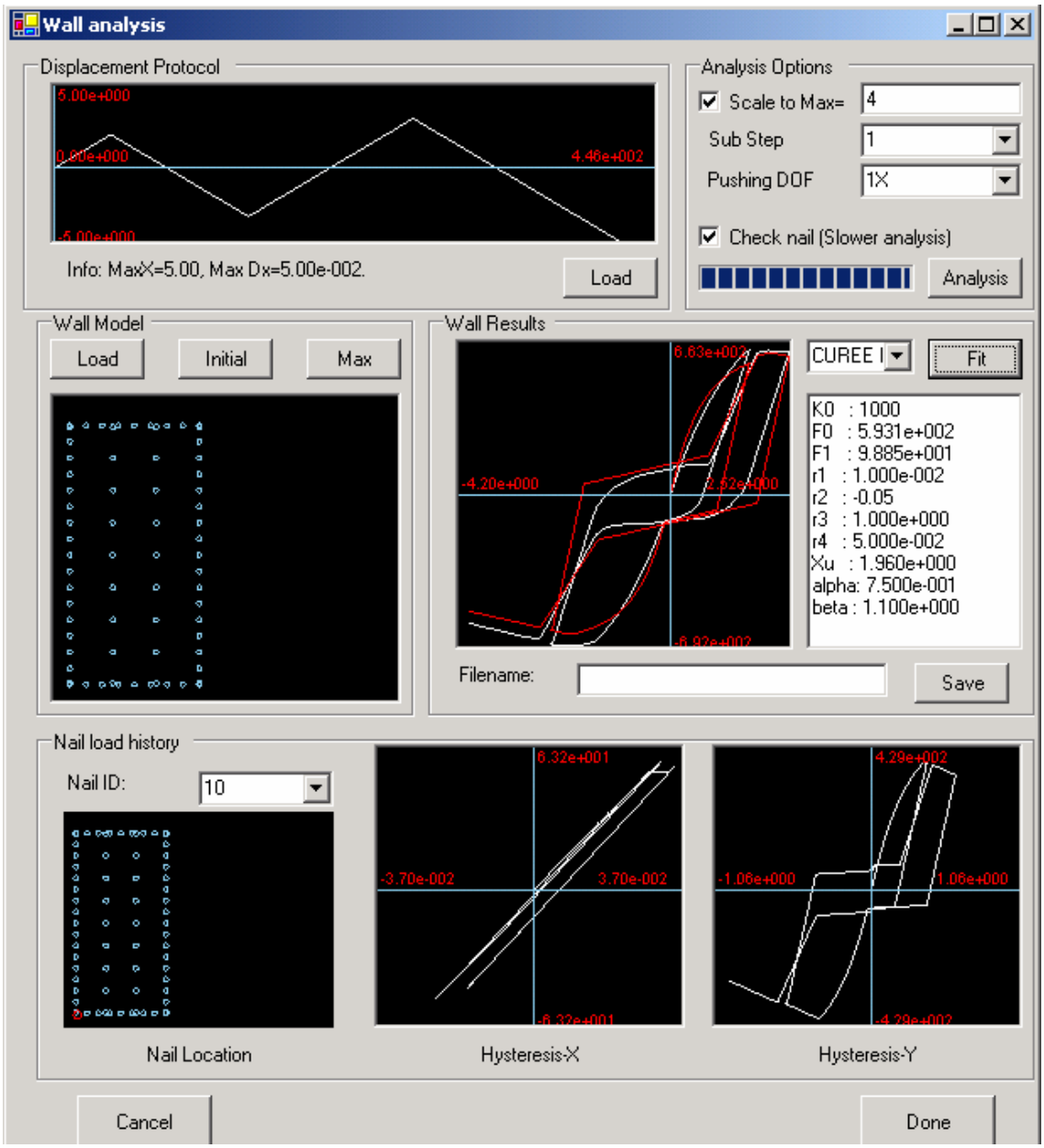


Figure 39. Obtain wall parameters from NP analysis

5.3 Woodframe structure loss analysis

This example only illustrates the steps to perform a vulnerability based loss analysis. The procedure to conduct Virtual Hazard simulation is very similar.

Step 1: Load the vulnerability based simulation tool box.
Step 2: Enter seed value as 1.
Step 3: Select structure quality as "Superior", then load model "Residential.sap".
Step 4: Set the number of models to 10, set the number of loss samples to 50.

Step 5: Load element information file "ResidentialElement.txt". The damage fragility input will be enabled and ID 1~4 will appear in the drag-down list.

Step 6: Select 1 for drag-down list, click on "Assign fragility", select component fragility file "Frag_Shearwall.txt". The loading will take a while, be patient. Repeat this process and load 2-"Frag_GWBdrywall.txt", 3-"Frag_DW.txt" (for doors and windows), and 4-"Frag_Contents.txt".

Step 7: Open earthquake hazard environment file "Eq_SanFrancisco.txt" file, you will see a list of earthquake files (*.eq2). Change the file path of these files to the eq2/ directory in your SAPWood installation folder. This is to make sure these files can be loaded from the right place. After that, save the file and load it to the tool box.

Step 8: Enter the intensity grid values. The array "0.05 0.1 …4" was used in this example.

Step 9: Enter the output name as "Vresults" and click on "Vulnerability analysis".

Step 10: The analysis may take several hours or as much as a day. After it is done, the output file "Vresults.SEC" can be found in the output directory, which contains all the simulated single earthquake loss data and the vulnerability model.

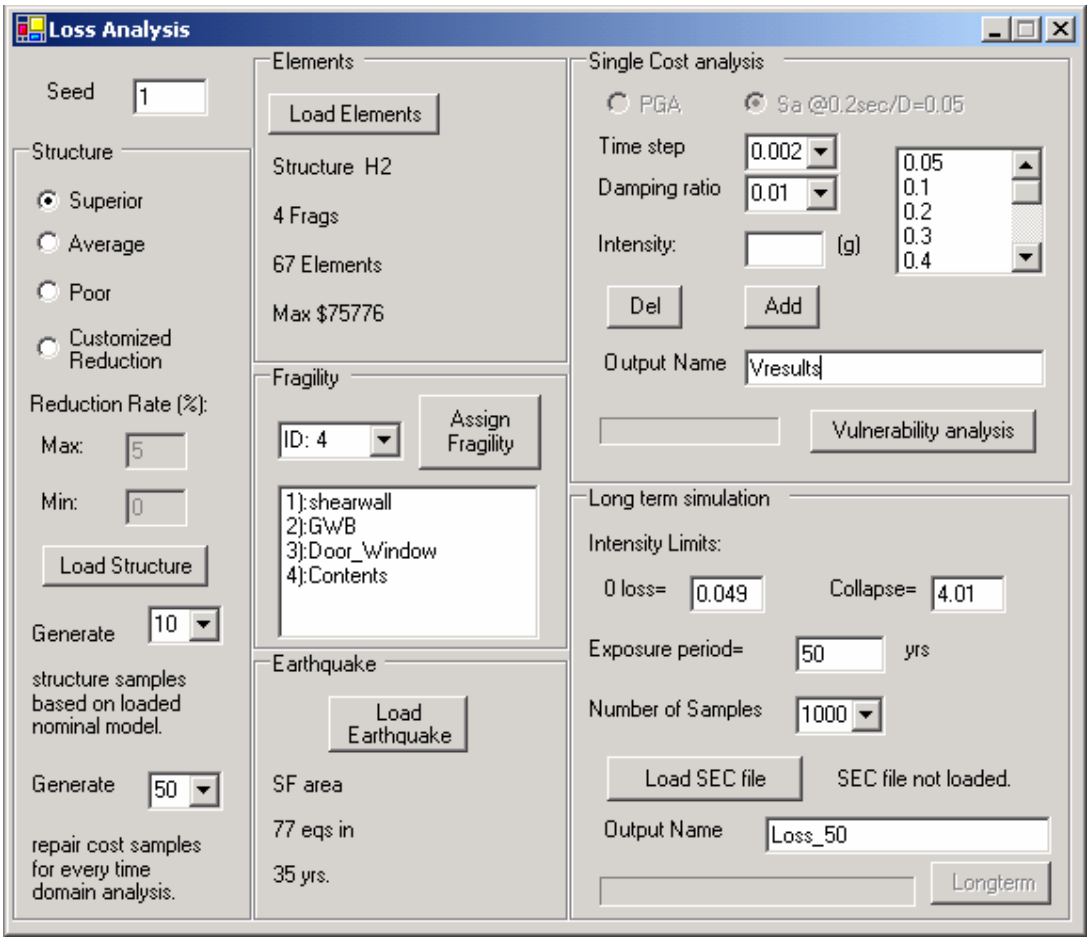


Figure 40. Loss simulation of the residential building

Step 11: Enter 0.049 and 4.01 as the zero-loss and collapse intensities. Enter 50 years as exposure period and choose 1000 samples.

Step 12: Load the "Vresults.SEC" file just generated.

Step 13: Use "Loss_50" as the output name and click "Long term". After the analysis, 1000 samples of 50 year economic loss of the structure in San Francisco area will be written to "Loss_50.LOS" file. This

simulation will only take a couple of minutes.
Step 14: Repeat Step 13 for different exposure periods needed.

One can see the benefit of vulnerability based simulation is that the time consuming vulnerability analysis for a structure only needs to be conducted once. Then the long term simulation will only need the SEC file and can be done very quickly.

References

1. Pei S., and van de Lindt J.W. Methodology for earthquake-induced loss estimation: An application to wood-frame buildings. Accepted by *Structural Safety* 2008, in Press.

2. Folz, B. & Filiatrault, A. Seismic analysis of woodframe structures. I: Model formulation. Journal of Structural Engineering 2004; 130(9): 1353-1360.

3. Folz, B. and Filiatrault, A. Seismic analysis of woodframe structures. II: Model implementation and verification. Journal of Structural Engineering, 2004; 130(9): 1361-1370.

4. Pang, W.C., Rosowsky, D.V., Pei, S. & van de Lindt, J.W. Evolutionary parameter hysteretic model for wood shearwalls. Journal of Structural Engineering, 2007; accepted in press.

5. Folz, B. & Filiatrault, A. Cyclic analysis of wood shear walls. Journal of Structural Engineering, 2001; 127(4): 433-441.

6. van de Lindt, J.W., Rosowsky, D.V., Filiatrault, A., Symans, M.D. & Davidson, R.A. The NEESWood project: Progress on the development of a performance-based seismic design philosophy for mid-rise woodframe construction. 9th World Conference on Timber Engineering, Portland, OR. 2006.

7. Christovasilis, I.P., Filiatrault, A., and Wanitkorkul, A. (2007). "Seismic testing of a full-scale two-story wood light-frame building: NEESWood benchmark test" Neeswood Report No. NW-01, Dept. of Civil, Structural and Environmental Engineering, University at Buffalo, State University of New York, Buffalo, available on-line at http://www.engr.colostate.edu/NEESWood/.

8. J.W. van de Lindt, S. Pei, H. Liu, and A. Filiatrault. (2010) "Three-Dimensional Seismic Response of a Full-Scale Light-Frame Wood Building: A Numerical Study", ASCE Journal of Structural Engineering. 136(1), 56-65.

9. S. Pei and J.W. van de Lindt. (2009). "Coupled shear-bending formulation for seismic analysis of stacked wood shear wall systems" Earthquake Eng. and Structural Dynamics, 38: 1631-1647.

10. S. Pei and J.W. van de Lindt. (2009) "Systematic seismic design for manageable loss in woodframe buildings", Earthquake Spectra, 25(4), 851-868.

11. S. Pei, and J.W. van de Lindt. (2009). "Methodology for earthquake-induced loss estimation: An application to woodframe buildings." Structural Safety, 31, 31-42.

12. J.W. van de Lindt, S. Pei, and H. Liu (2008). "Performance-Based Seismic Design of Wood Frame Buildings using a Probabilistic System Identification Concept" ASCE Journal of Structural Engineering, 134(2), 240-247.

13. S. Pei (2007). "Loss analysis and loss based seismic design for woodframe structures", Ph.D. Dissertation, December 2007, Colorado State University, 243 pages.

Appendix

A. Input file formats

The user might find it easier to create model files (biaxial model, NP model) with text editor rather than using the model builders. Or the user might need to make modifications on the model file created by model builders or other users. In order to provide information on the structure for the model files, detailed explanations for the *.SAP file and *.NPS file are listed in this section for your references.

Biaxial model file format (*.SAP)

Each data input in *.SAP file should be separated by a space

Please also see the example (Benchmark.SAP) distributed with the installation package for details.

Location (Row #)	Data	Description
1	N	Number of stories
2	Wn1 Wn2 … WnN	Number of wall in each story
3	Wmass(1) Wmass(2) …	Wall concentrated masses in story 1
…	…	…
2+N	Wmass(1) Wmass(2) …	Wall concentrated masses in story N
2+N+1	Wrmass(1) Wrmass(2) …	Wall rotational masses in story 1
…	…	…
2+2N	Wrmass(1) Wrmass(2) …	Wall rotational masses in story N
2+2N+1	Wtype(1) wtype(2) …	Wall type[1] in story 1
…	…	…
2+3N	Wtype(1) wtype(2) …	Wall type in story N
2+3N+1	Wdir(1) wdir(2) …	Wall direction[2] in story 1
…	…	…
2+4N	Wdir(1) wdir(2) …	Wall direction in story N
2+4N+1	WlocX(1) wlocX(2) …	Wall location X in story 1
…	…	…
2+5N	WlocX(1) wlocX(2) …	Wall location X in story N
2+5N+1	WlocY(1) wlocY(2) …	Wall location Y in story 1
…	…	…
2+6N	WlocY(1) wlocY(2) …	Wall location Y in story N
2+6N+1	Wall Parameters	Wall parameters for Wn1 walls in story 1, each one use a single row.
…	…	
2+6N+Wn1	Wall Parameters	
…	…	…
2+6N+Wt-WnN+1	Wall Parameters	Wall parameters for WnN walls in story 1,

…	…	each one use a single row.
2+6N+Wt[3]	Wall Parameters	
2+6N+Wt+1	0	Coordinate option[4]
3+6N+Wt+1	Node1X Node2X …	X coordinates for Nodes in the 1st story diaphragm
3+6N+Wt+2	Node1Y Node2Y …	Y coordinates for Nodes in the 1st story diaphragm
…	…	…
3+8N+Wt-1	Node1X Node2X …	X coordinates for Nodes in the Nth story diaphragm
3+8N+Wt	Node1Y Node2Y …	Y coordinates for Nodes in the Nth story diaphragm
3+8N+Wt+1	Dmass1 Damss 2 …	Distributed mass for each story diaphragm
3+8N+Wt+2	CmassN1[5] CmassN2 …	Number of Concentrated mass for each story
5+8N+Wt+1	X Y M RM	Location, Mass, and Rotational Mass for Concentrated mass #1 at story 1
…	…	…
5+8N+Wt+Ct[6]	X Y M RM	Location, Mass, and Rotational Mass for Concentrated mass #CmassNN at story N
5+8N+Wt+Ct+1	Wlen(1) wlen(2) …	Wall length for story 1
…	…	…
5+9N+Wt+Ct	Wlen(1) wlen(2) …	Wall length for story N
5+9N+Wt+Ct+1	Rp(1) Rp(2) …	Wall exterior nail spacing for story 1
…	…	…
5+10N+Wt+Ct	Rp(1) Rp(2) …	Wall exterior nail spacing for story N
5+10N+Wt+Ct+1	Whigh(1) Whigh(2) …	Wall height for story 1
…	…	…
5+11N+Wt+Ct	RWhigh(1) Whigh(2) …	Wall height for story N
5+11N+Wt+Ct+1	Rp(1) Rp(2) …	Reserved wall parameter[7] for story 1
…	…	…
5+12N+Wt+Ct	Rp(1) Rp(2) …	Reserved wall parameter for story N
5+12N+Wt+Ct+1	Rp(1) Rp(2) …	Reserved wall parameter for story 1
…	…	…
5+13N+Wt+Ct	Rp(1) Rp(2) …	Reserved wall parameter for story N
5+13N+Wt+Ct+1	Rp(1) Rp(2) …	Reserved wall parameter for story 1
…	…	…
5+14N+Wt+Ct	Rp(1) Rp(2) …	Reserved wall parameter for story N

[1] Wall type ID: 1- Linear; 2- Bilinear; 3- CUREE 10-parameter hysteresis; 4- EPHM (16 parameter)

[2] Wall direction ID: 1- in X; 2- in Y.

[3] Wt= total number of walls in the model.

[4] Coordinate option: can be 0 or 1, default value is 0. Please see FAQ for more information.

[5] If there is no concentrated mass, this row still needs to be inputted with 0's, the number of 0 should be equal to the number of stories. If this row is 0, the concentrated mass information will not be entered.

[6] Ct= total number of concentrated masses in the model (NOT include the concentrated mass associated with the

wall).

[7] Reserved wall parameters are for future upgrade options for wall elements. They will not be used in the analysis of the current version. However, they must be input in the correct format so that the file will load properly (no need to worry about them when using the model builder). Use 1 for all of these values, and use a correct number of them (should be equal to the wall number in each story).

Triaxial model file format (*.SAP)

Triaxial model can be constructed by adding vertical spring (and possibly FPS isolators) information to the biaxial model file. The format of triaxial model is explained in the format instruction excel file distributed together with this package.

Shear Building model file format (*.sbd)

Each data input in an *.sbd file should be separated by a space.

Location (Row #)	Data	Description
1	N	Number of story (N)
2	Mass(1) … Mass(N)	Mass of each story
3	Type(1) … Type(N)	Spring type of each story
4	Parameter for Spr 1	Spring parameters for each story
…	…	
3+N	Parameter for Spr N	
4+N	Reserved space	Reserved for future use, input -1 here for each story for now

SAPWood-NP model file format (*.NPS)

Each data input in an *.NPS file should be separated by a space. Every coordinate value in the model is based on a global coordinate system defined by the user at the beginning of the model construction.
Please also see the example (mywall.NPS) distributed with the installation package for details.

Location (Row #)	Data	Description
1	CN NN	Number of components (CN) and Nail lines (NN)
2	X(1) Y(1)	Center location for each component, one row for one component.
…	…	
1+CN	X(CN) Y(CN)	
1+CN+1	Dof1 Dof2 …	ID for Dof's need to be fixed[1]
2+CN+1	Ntype p1 p2 ….	Nail type (Ntype) followed by parameters (p1 p2 …) for nail lines
…	…	
2+CN+NN	Ntype p1 p2 ….	
2+CN+NN+1	Xs Ys Xe Ye Sp dir C1 C2 n	Start point (Xs Ys) end point (Xe Ye) spacing (Sp) Direction (dir) Component 1(C1) component2 (C2) Nail number (n) for nail lines[2].
…	…	
2+CN+2NN	Xs Ys Xe Ye Sp dir C1 C2 n	

[1] If one want to fix the Nth component, the Dof ID will be N*3-2 N*3-1 N*3, since the NP model assign 3 Dof's for

each component.

[2] Direction ID for NP model is: 1- in X; 2- in Y; 3- depends on start and end points. One could leave Nail number n to be -1 to let the program decide the nail number based on the spacing and start/end points. If you want to put just 1 nail in a certain location, use that location as a start point, use -1 for spacing and put 1 in nail number.

Wall configuration input file format

Wall configuration file can be used to input multiple wall configurations in the wall parameter database tool box and generate wall parameters all at once. It can just be a *.txt file containing the right format and information. Each data input in this file should be separated by a space.

Location (Row #)	Data	Description
1	N	Number of walls (N) in the file
2	Type Nail Pattern Height Length[1]	Wall configuration information for each wall in the file
…	…	
1+N	Type Nail Pattern Height Length	

[1] Wall configurations are represented by integer numbers as follows: Type (1-Drywall, 2-Shearwall), Nail (1-8d common, 2-10d common), Pattern (1-2/12, 2-3/12, 3-4/12, 4-6/12, 5-12/12), Height (1-8ft, 2-9ft, 3-11ft), Length (use the actual length of the wall in unit of inches).

Element information file

Element information file can be used to indicate the information about all damageable components in the structure during loss analysis. It can just be a *.txt file containing the right format and information. Each data input in this file should be separated by a space.

Location (Row #)	Data	Description
1	File description	A one line comment of the file contents
2	Ne Nf	Number of elements (Ne) in the structure; Number of Fragility types (Nf) in the structure.
3	StID WID ResID Ft Rc[1]	The structural element (story number StID and wall number WID) associated with each cost-inducing element, the response type (ResID), fragility type (Ft), and replacement cost (Rc).
…	…	
2+Ne	StID WID ResID Ft Rc	
3+Ne	Description for element 1	Each component has a one line description.
…	…	
2+2*Ne	Description for element Ne	

[1] Use story number for each story as StID, use 0 for the ground (some damageable components need to be associated with ground acceleration, such as furniture on the bottom floor). Use wall number in the structural model as the WID if the damage is associated with the response of a wall, use -1 to indicate the story response. Use 1 for ResID for wall drift, -1 for story drift, -2 for story acceleration. Ft is the type of fragility applied to the component, 1<=Ft<=Nf. RC is the cost to replace the component. It is used to indicate the maximum loss a component could induce.

Hazard environmental file

Hazard environmental file can be used to indicate the information about earthquake hazard in the structure during loss analysis. It can just be a *.txt file containing the right format and information. Each data input in this file should be separated by a space.

Location (Row #)	Data	Description
1	Name of the location	The location for the hazard character this file represents
2	Ne	Number of earthquake records
3	Location of the first earthquake record file	The location of *.eq2 files used to represent the uncertainty of earthquake ground motion. The location must include the full path information.
…	…	
2+Ne	Location of the last earthquake record file	
3+Ne	Tn Damp IsCDF[1]	Intensity information
4+Ne	List of Intensities[2]	Intensity values on the hazard curve
5+Ne	List of Annual PE or CDF value	Probability values on the hazard curve
6+Ne	Nc Time	Number of earthquake occurrence (Nc) in a given period (Time) in years

[1] Tn is the natural period if Spectral Acceleration is used, use negative Tn indicates the intensity type is PGA. Damp is the damping ratio for Sa. IsCDF indicates the type of probability indicator used in the hazard curve. The value 1 means the CDF value is used. Other values means the annual probability of exceedance is used.

[2] List of intensities should be in ascending order. When paired with the next row of probability values, it should provide probabilistic hazard information of the intensity distribution on the building site. The Intensity-annual PE curve data can be obtained directly from USGS website for locations in U.S.

Component fragility file

Component fragility file can be used to indicate the information and data to construct damage fragility distributions for each type of component in the structure during loss analysis. It can just be a *.txt file containing the right format and information. Each data input in this file should be separated by a space. All component fragility files in SAPWood are based on a 4-level damage system: Level 1-Non-detectible, level 2-Repairable, Level 3-Borderline repairable, and Level 4-Replace/demolish.

Location (Row #)	Data	Description
1	Name of the Fragility type	One line comment of the damageable component this file represents
2	Rlow Rhigh Rn	The lower (Rlow) and upper (Rhigh) boundaries define the range for the response[1] distribution given damage. This range will be divided into Rn segments and the PDF values will be calculated on each segment. Rn is the resolution of the response

		distribution.
3	Cn Pn	Cn is the resolution of the Cost distribution, Pn is the resolution of Prior distributions
4	MuA MuB StdA StdB Nd	The boundaries[2] for uniform priors for Response distribution given damage level 1 (denoted by MuA MuB StdA StdB). Nd is the number of lines for the data used to update the Bayesian model
5~4+Nd	The response data given damage level 1	Multiple lines of response data given damage level 1.
…	…	…
Various[3]	MuA MuB StdA StdB Nd	The boundaries for uniform priors for Response distribution given damage level 4 (denoted by MuA MuB StdA StdB). Nd is the number of lines for the data used to update the Bayesian model
	The response data given damage level 4	Multiple lines of response data given damage level 4.
	MuA MuB StdA StdB Nd	The boundaries for uniform priors for normalized cost[4] distribution given damage level 1 (denoted by MuA MuB StdA StdB). Nd is the number of lines for the data used to update the Bayesian model
	The cost data given damage level 1	Multiple lines of cost data given damage level 1.
	…	…
	MuA MuB StdA StdB Nd	The boundaries for uniform priors for normalized cost distribution given damage level 4 (denoted by MuA MuB StdA StdB). Nd is the number of lines for the data used to update the Bayesian model
	The cost data given damage level 4	Multiple lines of cost data given damage level 4.

[1] Currently the responses used in SAPWood loss analysis are mainly maximum inter-story drift (Drift/Story Height, dimensionless) or maximum acceleration (use unit of gravity).

[2] Priors for the mean and standard deviation of the predictive distribution is uniform distributions with boundaries [MuA MuB] and [StdA StdB].

[3] The row number of later inputs depends on the amount of data inputted in previous lines. For example, the response data under damage level 1 might be only 1 line, but data for level 2 might be 4 lines, etc. So a definite number could not be given here.

[4] Normalized cost is used here so the actual cost must be divided by the maximum possible cost (replacement cost) of the component first to be used at cost data. In this way, the cost distribution will always be between [0 1]. This enables the use of the same Fragility information on same type of component with different actual value, for example, two shearwalls of different construction cost. The cost fragility will be combined with the actual cost in the element information file to yield the real cost during simulation.

B. Output file formats

There are a number of output files generated from SAPWood. The ability to generate these output files provides the user with the option of post-processing the numerical analysis results with the software package of their choice. Most output files have very clear descriptions with the data so that it is easy to read and understand these files. The formats of these output files are illustrated with examples in this section.

- *.gen file: Generated by single earthquake excitation analysis. It provides general information about the time domain analysis results

Example file:

```
General information of the analysis 12/6/2007 11:13:38 AM :
Unit system (US): inch,lb,sec

-------------------------------------------------------
Earthquake input information:
Earthquake scaled with PGA during this analysis.
Earthquake input record(s):
      C:\Program Files\CSU\SAPWood\Earthquakes\Acm2.eq2PGA-X=0.34, PGA-Y=0.25
      EqX applied to model-X axis and EqY applied to model-Y axis.

-------------------------------------------------------
--------------------Response results--------------
Structure: SAWS House with 2 story.

Maximum responses---------------------------
Displacements:     X                  Y              R
Story--1: 8.069202e-001   3.115652e-001   1.993993e-003
Story--2: 9.093823e-001   3.905857e-001   2.135139e-003

Maximum responses---------------------------
Inter-story Drifts:     dX                 dY             dR
Story--1: 8.069202e-001   3.115652e-001   1.993993e-003
Story--2: 3.190663e-001   8.647249e-002   1.744782e-003

Maximum story forces---------------------------
Total force at story:     Fx                  Fy               Mr
Story--1: 2.442862e+004   2.461310e+004   2.428952e+006
Story--2: 2.304419e+004   1.100866e+004   5.801013e+006

Maximum story Accelerations---------------------------
Global Accel at story Centroid:     Ax                  Ay               Ar
Story--1: 2.413558e+002   1.164926e+002   3.346475e+000
```

Story--2: 2.336602e+002 1.417761e+002 4.359007e+000

The average acceleration is calculated using a 0.2sec time interval

Note: If the specified time interval is smaller than the time interval of input excitation, the excitation time interval is used to calculate the average Accel.

Maximum story Average Accelerations----------

Global Accel at story Centroid:(average) Ax Ay Ar

Story--1: 6.037049e+001 2.040264e+001 4.344337e-001

Story--2: 6.403841e+001 3.491183e+001 6.115605e-001

- *.Glo file: Generated by single earthquake excitation analysis. It provides global response of the analysis (Force output will only be available if "Include Force" is checked).

Example file:

Global results of the analysis 8/22/2006 3:14:39 PM :

Unit system (US): inch,lb,sec

Displacement output----------

Time (sec)	Story1 X	y	R	Story2 X	y	R
2.0000e-002	4.693345e-004	2.881727e-004	-5.012285e-009	4.906954e-004	2.955567e-004	3.752839e-011
4.0000e-002	1.345850e-003	8.843146e-004	-7.231205e-008	1.610710e-003	9.824215e-004	1.554425e-009
6.0000e-002	2.207709e-003	1.576522e-003	-2.869087e-007	3.102659e-003	1.960484e-003	6.669078e-009
8.0000e-002	3.100269e-003	2.347804e-003	-6.634809e-007	4.848521e-003	3.271791e-003	-9.460180e-009
1.0000e-001	3.653253e-003	2.905428e-003	-1.090316e-006	6.109361e-003	4.587523e-003	-1.384288e-007

…..

Force output----------

2.0000e-002	5.565619e-002	1.830492e-002	5.944105e-001	2.438840e-003	3.691284e-004	-1.056048e-002
4.0000e-002	1.570302e-001	5.526635e-002	1.529525e+000	3.014629e-002	4.910357e-003	-5.924926e-002
6.0000e-002	2.563258e-001	9.792289e-002	1.996804e+000	1.016278e-001	1.922002e-002	1.520290e-001

……

- *.hys file: Generated by single earthquake excitation analysis. It provides hysteretic response of all the wall elements in the model

Example file:

Wall hysteresis results of the analysis 8/22/2006 3:14:39 PM :

Unit system (US): inch,lb,sec

The wall hysteresis----------

the walls in story-1

Time (sec)	Drift-Wall1	Force-wall1	Drift-Wall2	Force-wall2 ……
2.0000e-002	0.000000e+000	0.000000e+000	0.000000e+000	0.000000e+000 …
4.0000e-002	4.699718e-004	1.353313e-002	4.699718e-004	6.289870e-003 …
6.0000e-002	1.355044e-003	3.838059e-002	1.355044e-003	1.784774e-002 …

….

- *.ehy file: Generated by SDOF identification analysis. It is just the equivalent hysteresis data of the global DOF.

Example file:

0.0000e+000 0.0000e+000
0.0000e+000 0.0000e+000
-4.1200e-001 -3.7023e+003
-8.2400e-001 -4.9348e+003
-1.2360e+000 -5.4620e+003
-1.0207e+000 -3.1967e+003
-8.0543e-001 -1.3546e+003
-5.9014e-001 1.9288e+002
-3.7485e-001 5.6080e+002
-1.5956e-001 8.4247e+002
……

- *.IDA file: Generated by IDA analysis. It contains the IDA results for all global and local (wall) DOFs.

Example file:

Results of the IDA analysis 12/10/2007 9:42:47 AM :
The earthquake scaling was based on PGA ratio X:Y=0.11611:0.11611 (from original record).
Intensity array applied to Ax
Unit system (US): inch,lb,sec
The X-Y excitation has been applied to the X-Y direction of the model

Beginning of Global records----------
PGA(g): 0.2000 0.6000
Displacement Disp x----------
Story1: 6.3198e-001 1.5082e+000
Story2: 2.3093e+000 7.3424e+000
Displacement Disp y----------
Story1: 5.1288e-001 1.7589e+000
Story2: 1.6130e+000 4.2261e+000
Displacement Disp r----------
Story1: 9.1298e-003 2.9664e-002
Story2: 1.2090e-002 6.3164e-002

Inter-story Drift-Dx----------
Story1: 6.3198e-001 1.5082e+000
Story2: 1.8866e+000 6.0572e+000
Inter-story Drift-Dy----------
Story1: 5.1288e-001 1.7589e+000
Story2: 1.1690e+000 3.0333e+000
Inter-story Drift-Rr----------
Story1: 9.1298e-003 2.9664e-002
Story2: 1.1929e-002 4.5942e-002

Fx----------
Story1: 1.0554e+001 2.2117e+001
Story2: 7.2445e+000 7.7042e+000
Fy----------
Story1: 8.6412e+000 2.1903e+001
Story2: 1.1243e+001 1.1860e+001
Mr----------
Story1: 1.0122e+003 7.4563e+002
Story2: 9.5875e+002 4.9766e+002

Ax----------
Story1: 5.1127e+002 7.0022e+002
Story2: 4.1323e+002 4.3836e+002
Ay----------
Story1: 5.2268e+002 6.4689e+002
Story2: 6.3718e+002 5.8200e+002
Ar----------
Story1: 2.5985e+001 2.4299e+001
Story2: 3.5682e+001 1.8608e+001

Averaging time interval=0.2 sec.
Note: If the specified time interval is smaller than the time interval of input excitation,
the excitation time interval is used to calculate the average Accel.
Ax (average)----------
Story1: 9.6829e+001 1.0938e+002
Story2: 9.6180e+001 2.0094e+002
Ay (average)----------
Story1: 8.6042e+001 1.5960e+002
Story2: 8.7776e+001 1.9986e+002
Ar (average)----------
Story1: 3.4271e+000 3.7689e+000
Story2: 4.9558e+000 3.7875e+000
End of Global records----------

Beginning of local records----------
PGA(g): 0.2000 0.6000
The Maximum Drift of the elements----------
Wall1-1:6.3466e-001 1.8320e+000
……

- *.MDA file: Generated by Multi-case IDA analysis. It contains the IDA results for all global and local (wall) DOF for multi-cases (each row of IDA data corresponds to a different earthquake record).

Example file:

Results of the MIDA analysis 11/29/2007 2:46:56 PM :

Unit system (US): inch,lb,sec

Input information----------

Totally 2 models analyzed.

1: C:\Program Files\CSU\SAPWood\Biaxial Models\BenchMark.sap

2: C:\Program Files\CSU\SAPWood\Biaxial Models\Example1.sap

Totally 2 Earthquake records analyzed.

1: C:\Program Files\CSU\SAPWood\Earthquakes\Acm2.eq2

2: C:\Program Files\CSU\SAPWood\Earthquakes\Anor2.eq2

Totally 2 PGA points analyzed.

1.0000e-001 2.0000e-001

Damping Ratio=2.0000e-002

Time step used=1.0000e-002

Analysis using PGA.

Beginning of Global records----------

The structure: C:\Program Files\CSU\SAPWood\Biaxial Models\BenchMark.sap----------

Story:1----------

Story:1 of 2--Dx:

PGA: 0.10 0.20

1.0214e-001 2.0514e-001

1.0455e-001 2.6594e-001

Story:1 of 2--Dy:

PGA: 0.10 0.20

1.5948e-001 3.8225e-001

1.5678e-001 4.3710e-001

……

The structure: C:\Program Files\CSU\SAPWood\Biaxial Models\Example1.sap----------

Story:1----------

Story:1 of 1--Dx:

PGA: 0.10 0.20

2.8271e-001 5.2220e-001

2.0225e-001 5.0098e-001

Story:1 of 1--Dy:

PGA: 0.10 0.20

1.1717e-001 2.2426e-001

3.9490e-001 2.8928e-001

……

Beginning of Local records----------

The structure: C:\Program Files\CSU\SAPWood\Biaxial Models\BenchMark.sap----------

Wall1/13 of Story1 --Drift

PGA: 0.10 0.20

1.0117e-001 2.3026e-001

8.9403e-002 2.7300e-001

Wall1/13 of Story1 --Force

PGA: 0.10 0.20

2.8431e+000 5.9141e+000

2.3748e+000 6.1857e+000

Wall2/13 of Story1 --Drift

PGA: 0.10 0.20

1.0117e-001 2.3026e-001

8.9403e-002 2.7300e-001

……

The structure: C:\Program Files\CSU\SAPWood\Biaxial Models\Example1.sap----------

Wall1/41 of Story1 --Drift

PGA: 0.10 0.20

3.2416e-001 5.8030e-001

2.5920e-001 7.3752e-001

Wall1/41 of Story1 --Force

PGA: 0.10 0.20

6.7388e+002 9.3401e+002

5.7929e+002 1.0731e+003

……

- *.SEC file generated by vulnerability analysis. It contains the simulated single earthquake loss data at different intensity levels and the vulnerability model obtained from these data. The output file includes at the very beginning the vulnerability parameters for the vulnerability model developed by Shiling Pei in his doctoral dissertation work. Then the simulated loss data is also given for every single intensity value.

Example file:

Start:Saturday, August 25, 2007\8:12:56 PM End:Sunday, August 26, 2007\12:30:50 PM

24

Intensity	Mu-ln	Std-ln	Pr(0)	Pc	all-mean	all-std
0.05	4.6697	1.0911	0.9635	0	10.34781	259.9304
0.1	4.9725	1.2373	0.2605	0	364.3525	2055.563
0.2	6.3333	1.4942	0.0145	0	2174.413	5383.001
0.3	7.3753	1.4969	0.0004	0	4532.886	7012.437
0.4	8.0784	1.3967	0	0	7154.492	8558.841
0.5	8.5834	1.2799	0	0	9907.467	9885.059
0.6	8.8302	1.1775	0	0.0499	14205.59	15627.12
0.7	9.139	1.0722	0	0.05	16726.28	15731.35
0.8	9.3482	0.99056	0	0.05	18848.61	15896.82
0.9	9.5684	0.90958	0	0.05	21619.73	16163.02
1	9.6706	0.88322	0	0.1499	27769.87	20370.73
1.1	9.7177	0.83641	0	0.26	33114.03	22722.14
1.2	9.8396	0.79459	0	0.32	37161.96	22794.91
1.3	9.9314	0.78699	0	0.3898	41306.45	22602.47
1.4	9.9787	0.74552	0	0.4997	46165.45	22257.63
1.5	10.08	0.70318	0	0.4947	47088.39	21391.11

1.6 10.073 0.64654 0 0.6046 50923 20614.35

1.8 10.214 0.56334 0 0.69 55150.27 17949.03

2 10.334 0.53497 0 0.7444 57915.16 15501.34

2.2 10.377 0.49719 0 0.8451 61286.58 12294.96

2.6 10.557 0.39561 0 0.8546 62436.18 10113.58

3 10.609 0.37661 0 0.8997 63744.13 7978.702

3.5 10.694 0.32275 0 0.8997 64085.17 7070.185

4 10.685 0.28389 0 0.9497 65053.44 5211.519

Beginning of the data

-1 5.000000e-002

0.000000e+000 0.000000e+000 0.000000e+000 2.616651e+002 0.000000e+000 2.176811e+002 0.000000e+000 1.879598e+001 0.000000e+000 1.797382e+002 0.000000e+000 0.000000e+000 0.000000e+000 8.369978e+001 …

-1 1.000000e-001

2.245363e+002 4.504117e+002 2.105297e+002 1.886487e+002 3.911586e+002 8.795535e+002 8.660370e+002 7.890755e+001 7.501007e+001 5.766284e+002 4.909902e+002 2.851999e+002 1.886487e+002 3.911586e+002 …

…

-1 4.000000e+000

6.593336e+004 6.675259e+004 6.768540e+004 6.647659e+004 6.761297e+004 6.669388e+004 6.696110e+004 6.525899e+004 6.448476e+004 6.770790e+004 6.730953e+004 6.511413e+004 …

- *.LOS file generated by long term loss simulation procedures (vulnerability based and Virtual Hazard). It is just a text file with the long term loss samples, with no description in the file (so one needs to specify the file name clearly when creating the LOS file).

Example file:

0.000000e+000 1.944744e+003 0.000000e+000 0.000000e+000 1.042034e+002 6.902475e+002 2.586495e+003 1.057963e+003 2.770213e+003 0.000000e+000 0.000000e+000 0.000000e+000 0.000000e+000 8.749048e+002 9.685940e+003 0.000000e+000 0.000000e+000 …

C. Spring models

Linear spring

The most simple spring type with only one stiffness parameter.

Parameter:

K – stiffness

Input sequence in SAPWood:

K

Bilinear spring

The simplest nonlinear spring type with energy dissipation properties.

Parameters:

K0 – initial stiffness

Ky – stiffness after the yielding point

Dy – the displacement corresponding to the yielding point

Input sequence in SAPWood:

K0 Ky Dy

SAWS-type hysteretic spring (CUREE model)

The 10-parameter model used in SAWS and CASHEW program, can be used to model the lateral load-resistance behavior of wood shearwall components.

Parameters:

K0 – initial stiffness.

F0 – the resistance force parameter of the backbone.

F1 – pinching residual resistance force.

R1 -- the stiffness ratio parameter of the backbone, typically be a small positive value.

R2 -- the ratio of the degrading backbone stiffness to K0, typically be a negative value.

R3 -- the ratio of the unloading path stiffness to K0, typically close to 1.

R4 – the ratio of the pinching load path stiffness to K0, typically under 0.1.

Xu – the drift corresponding to the maximum restoring force of the backbone curve.

Alpha – stiffness degradation parameter, usually takes a value between 0.5~0.9.

Beta – strength degradation parameter, usually takes a value between 1.01~1.5.

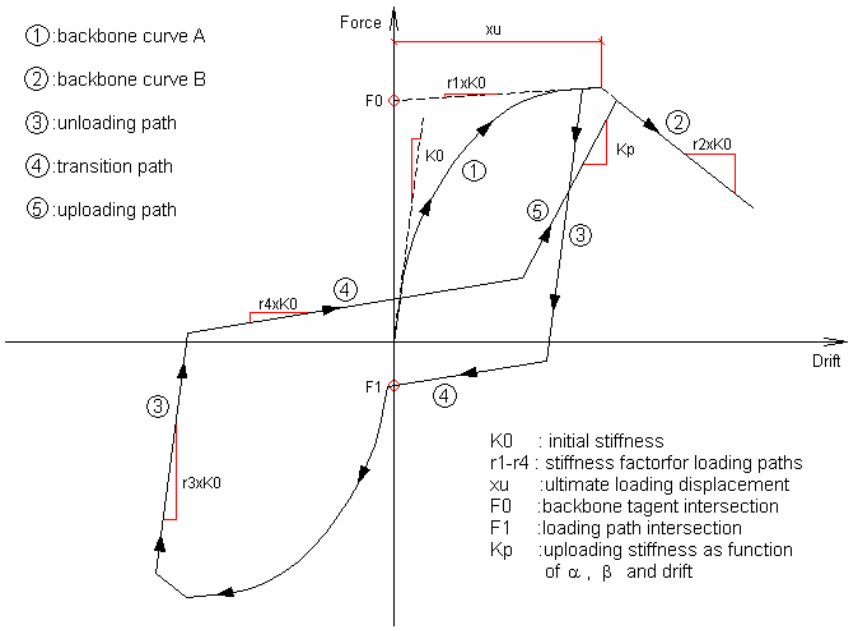


Figure 41. Loading paths and parameters in SAWS hysteresis model

Input sequence in SAPWood:

K0 F0 F1 r1 r2 r3 r4 Xu Alpha Beta

Evolutionary Hysteretic Parameter Model (EPHM) hysteresis

The 16-parameter hysteretic model developed at CSU. It allows more adaptive modeling of the degradation behavior of the wood shearwall components. It is recommended to model shearwalls with the EPHM when they might experience significant damage (large drift level) during the earthquake.

Parameters:

K0 – initial stiffness

F0 -- the resistance force parameter of the backbone.

R1 -- the stiffness ratio parameter of the backbone, typically be a small positive value.

Xu – the drift corresponding to the maximum restoring force of the backbone curve.

R2 -- the ratio of the degrading backbone stiffness (linear portion) to K0, typically be a negative value.

Xu1 – the drift corresponding to the end of the linear portion degrading backbone, the backbone degrades exponentially after this point.

P1 – the exponential degrading rate parameter of the backbone, b in Fig. 30.

F1m – maximum value that the residual pinching force can reach. A local parameter for evolutionary parameter FI (residual pinching force).

F1r – minimum value that the residual pinching force in severe damage stage. A local parameter for evolutionary parameter FI.

DF1a – tracking damage index (average maximum drift) corresponding to the starting point of the plateau portion of the FI degrading function. FI varies linearly before this point.

DF1b -- tracking damage index (average maximum drift) corresponding to the ending point of the plateau portion of the FI degrading function. FI degrades exponentially after this point.

pF1 – the exponential degrading rate parameter of the FI degradation function.

pr4 -- the exponential degrading rate parameter of the KI (the tangent stiffness at the point where loading paths intersects with Y axis) degradation function.

r4r – ratio of the residual KI value to initial stiffness.

Beta -- strength degradation parameter, usually takes a value between 1.01~1.5.

Fur – residual resistance force value of the backbone in severe damage stage.

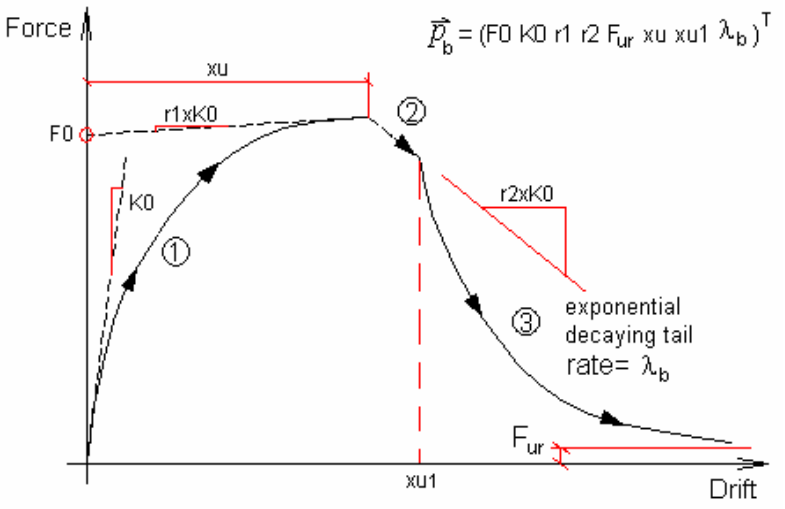


Figure 42. Backbone curve for EPHM hysteresis

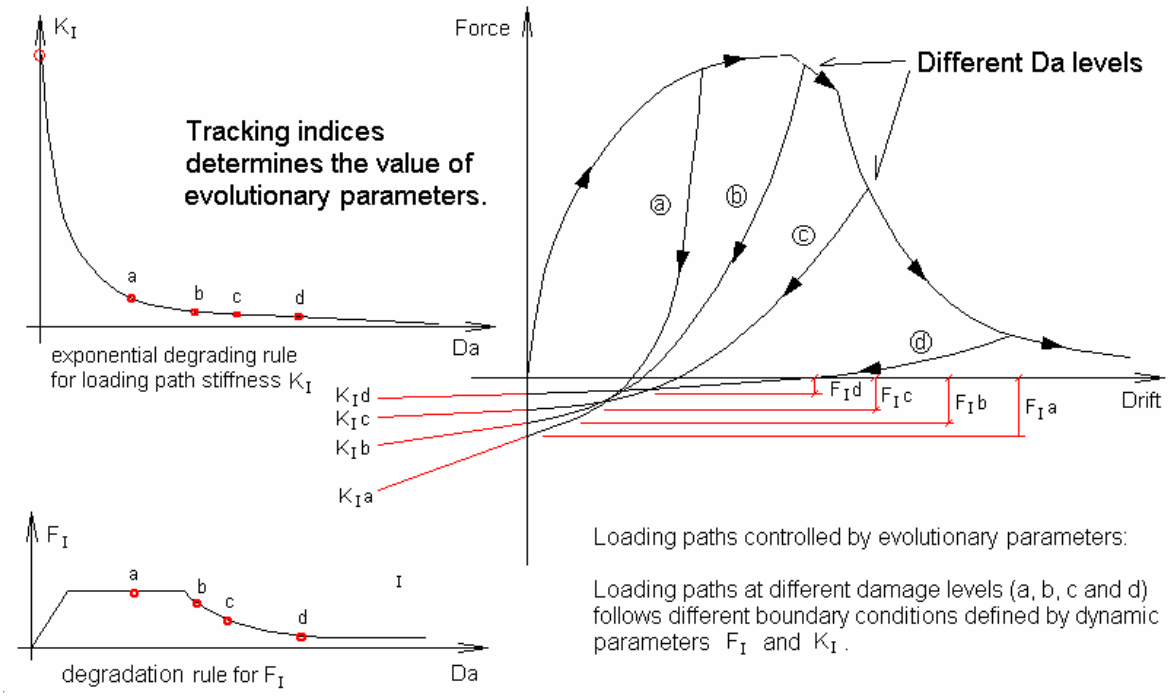


Figure 43. Degradation of loading paths

Input sequence in SAPWood:

K0 F0 r1 Xu r2 Xu1 P1 F1m F1r DF1a DF1b pF1 pr4 r4r Beta Fur

Special picewise linear vertical spring

The vertical spring is designed to represent the behavior of the vertical resistance between wood diaphragm floors provided by the combination of hold-down hardware and compression studs. The parameters for the spring and its behavior were shown in Figure 44 below.

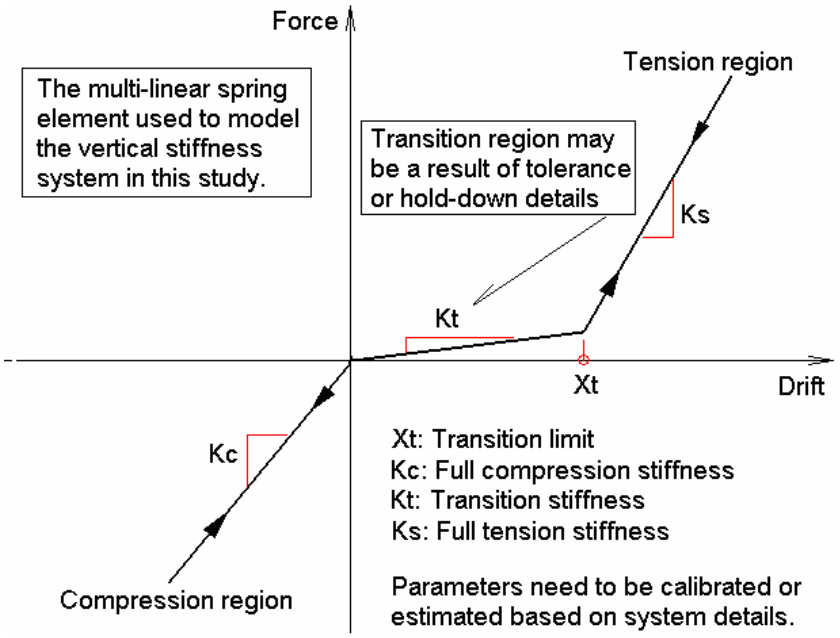

Force
Tension region
The multi-linear spring element used to model the vertical stiffness system in this study.
Transition region may be a result of tolerance or hold-down details
Ks
Kt
Xt
Drift
Kc
Xt: Transition limit
Kc: Full compression stiffness
Kt: Transition stiffness
Ks: Full tension stiffness
Compression region
Parameters need to be calibrated or estimated based on system details.

Figure 44. Special spring for vertical elements

D. Example structure configuration

One of the structures used in the example section of this manual is based on the benchmark house structure for the NEESWood project (report NW-01 by Christovasilis et al., available from the NEESWood website). The model file "Benchmark.sap" can be found in the installation folder after SAPWood is installed. This model is a simplified model in which only part of the shearwalls are considered. There's no concentrated mass in this model and the mass is assumed to be uniformly distributed over the floor diaphragm.

The coordinate system, floor plan, and the location/numbering of the wall elements are illustrated in Fig. 45 and 46.

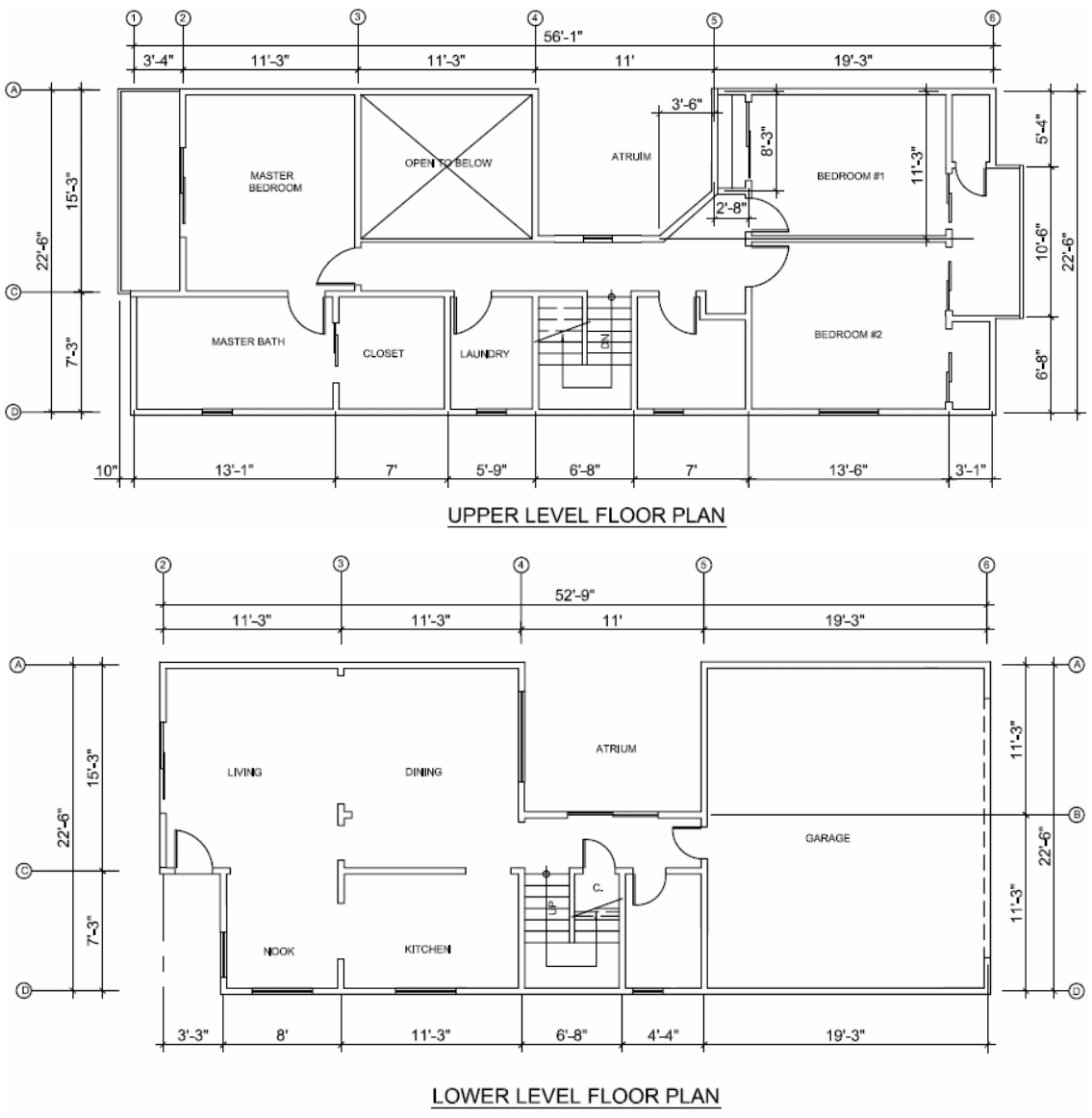


Figure 45. Floor plans for the benchmark structure

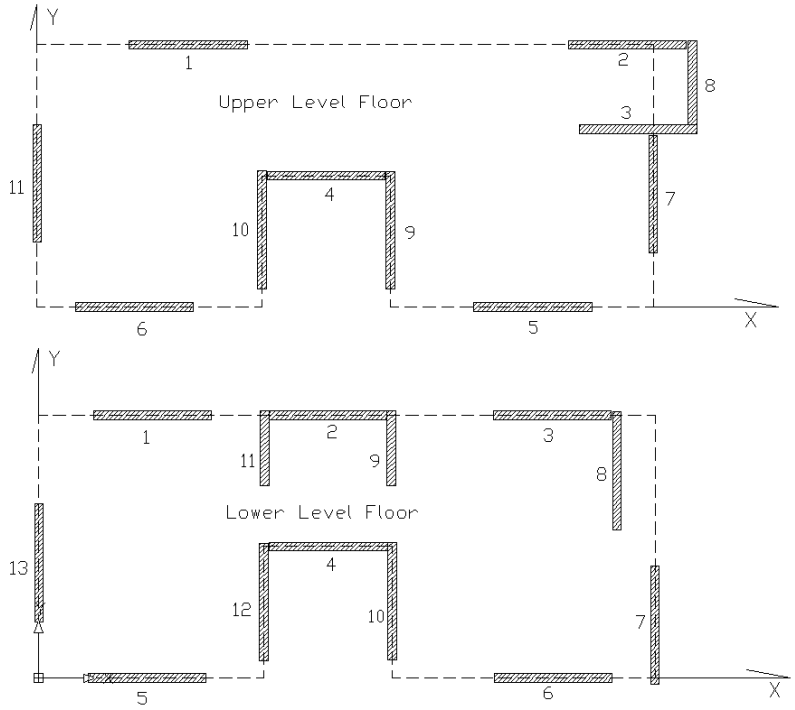


Figure 46. Wall element layout in the simplified model

The structure used in loss analysis example is a typical two-story woodframe single family home. The floor plan is illustrated in Figure 47, with red lines denote the drywall partition walls. The model file "Residential.sap" can be found in the installation folder after SAPWood is installed.

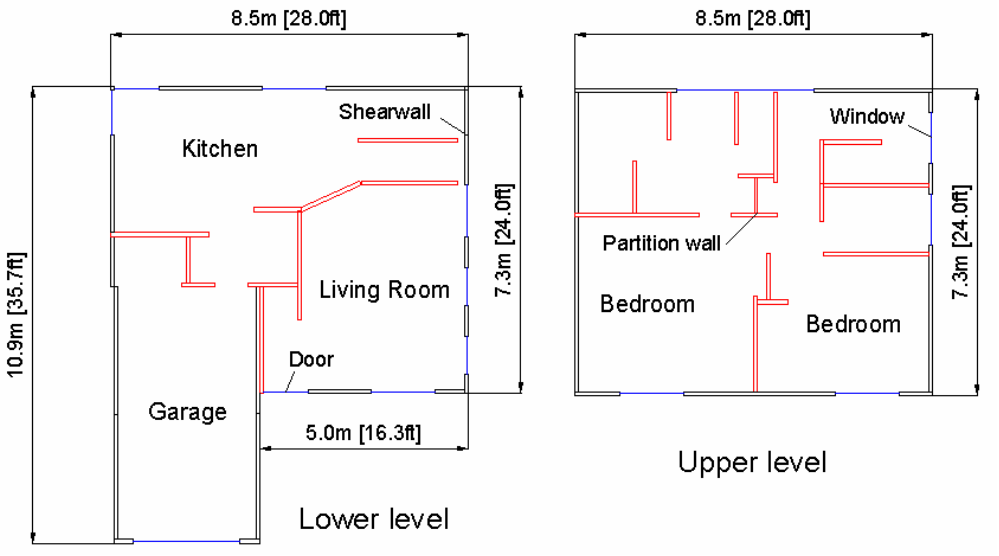


Figure 47. Residential structure floor plan